

Board Station for New Users Training Series

Module 3: Creating PCB Geometries

Software Version 8.5_2



Copyright © 1991-1996 Mentor Graphics Corporation. All rights reserved.
Confidential. May be photocopied by licensed customers of
Mentor Graphics for internal business purposes only.

The software programs described in this document are confidential and proprietary products of Mentor Graphics Corporation (Mentor Graphics) or its licensors. No part of this document may be photocopied, reproduced or translated, or transferred, disclosed or otherwise provided to third parties, without the prior written consent of Mentor Graphics.

The document is for informational and instructional purposes. Mentor Graphics reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Mentor Graphics to determine whether any changes have been made.

The terms and conditions governing the sale and licensing of Mentor Graphics products are set forth in the written contracts between Mentor Graphics and its customers. No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Mentor Graphics whatsoever.

MENTOR GRAPHICS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OR MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

MENTOR GRAPHICS SHALL NOT BE LIABLE FOR ANY INCIDENTAL, INDIRECT, SPECIAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING BUT NOT LIMITED TO LOST PROFITS) ARISING OUT OF OR RELATED TO THIS PUBLICATION OR THE INFORMATION CONTAINED IN IT, EVEN IF MENTOR GRAPHICS CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

RESTRICTED RIGHTS LEGEND Use, duplication, or disclosure by the Government is subject to restrictions as set forth in the subdivision (c)(1)(ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013.

A complete list of trademark names appears in a separate "Trademark Information" document.

Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.

This is an unpublished work of Mentor Graphics Corporation.

TABLE OF CONTENTS

About This Trainingxv

Workbook Organization	xv
Related Documentation	xv
Documentation Conventions.....	xv
Installation Procedure	xv

Lesson 1

Creating Design Geometries 1-1

Objectives.....	1-2
Board Station Terminology	1-3
Basic Geometry Types.....	1-4
Pin and Via Padstack Geometries	1-5
Component Geometry	1-6
Board Geometry	1-7
Generic Geometry.....	1-8
Display Layers Categories	1-9
Data Reporting Layers.....	1-10
Data Storage Layers	1-11
Electrical Layers	1-13
Top/Bottom Mapping Layers	1-15
User-Defined Layers	1-18
Containers and Design Objects	1-20
Library Management	1-21
Library Management System	1-21
Default Directory Hierarchy.....	1-22
Using the Default Directory Hierarchy	1-25
Libraries	1-26
Saving Geometries	1-27
Lab Exercise	1-29

TABLE OF CONTENTS [Continued]

Lab 1

Introduction to LIBRARIAN 1-31

Introduction	1-31
Procedure	1-31
Preparation for Lab	1-32
Setting Up to Create the Geometry Logo	1-34
Creating the Logo Border.....	1-36
Creating the Logo Text.....	1-41
Adding the Shapes to the Logo	1-43
Reading and Modifying Geometries	1-45
Saving Geometries and Leaving LIBRARIAN.....	1-54

Lesson 2

Basic Geometry Creation Techniques 2-1

Objectives.....	2-2
Setting up the Editing Environment	2-3
Edit Layers.....	2-3
View Layers.....	2-5
The Display Grid	2-6
Grid Visibility and Grid Snapping.....	2-7
Selecting Geometry Data	2-7
Unselecting Geometries	2-8
Select Filter and Unselect Filter	2-9
Coordinate Systems	2-10
Absolute Coordinate System.....	2-10
Delta Coordinate System and the Basepoint.....	2-11
Snapping	2-13

TABLE OF CONTENTS [Continued]

Lesson 2

Basic Geometry Creation Techniques (Continued)

Creating Geometries	2-15
Adding Lines	2-15
Adding Polygons	2-18
Adding Circles.....	2-18
Adding Arcs.....	2-20
Deleting Geometries	2-21
Moving and Copying Geometries	2-21
Lab Exercise	2-26

Lab 2

Basic Geometry Creation Techniques.....2-27

Introduction	2-27
Procedure	2-27
Preparation for Lab	2-28
Creating a Generic Geometry	2-29

Lesson 3

Creating Component Geometries 3-1

Objectives.....	3-2
Creating New Geometries.....	3-2
Changing Generic Geometries.....	3-4
Attributes.....	3-7
Pin Padstack Attributes.....	3-8
Via Padstack Attributes.....	3-10
Pin/Via Padstack Attribute.....	3-11

TABLE OF CONTENTS [Continued]

Lesson 3

Creating Component Geometries (Continued)

Component Attributes..... 3-12

Adding Pins 3-16

Checking Geometries 3-17

Lab Exercise 3-18

Lab 3

Creating Component Geometries 3-19

Introduction 3-19

Procedure 3-19

 Preparation for Lab 3-20

 Building the Pin Padstacks..... 3-21

 Building Via Padstack Geometries 3-24

 Building Components Geometries 3-25

 Saving Geometries and Leaving LIBRARIAN..... 3-49

Lesson 4

Creating Board Geometries..... 4-1

Objectives..... 4-2

Process for Creating a Board 4-2

Starting a Design 4-3

Board Attributes..... 4-4

More Board Attributes 4-8

Other Board Features 4-10

Design Rules 4-10

Physical Layers 4-12

Default Physical Layers..... 4-13

Lab Exercise 4-14

TABLE OF CONTENTS [Continued]

Lab 4

Creating Board Geometries.....	4-15
Introduction	4-15
Procedure	4-15
Preparation for Lab	4-16
Creating the Board Geometry.....	4-18
Setting Default Design Rules.....	4-42
Checking the Board Geometry.....	4-52
Saving Geometries and Leaving Librarian	4-53

Lesson 5

Mapping and Catalog Files.....	5-1
Objectives.....	5-2
Overview	5-2
Mapping Files	5-4
Comp Property.....	5-7
Non-homogeneous Mapping File	5-8
Common Pins.....	5-9
Pinsets	5-10
Catalog Files.....	5-16
Catalog Properties.....	5-20
User Comments in Catalog Files.....	5-20
Catalog Libraries.....	5-21
Default Directory Hierarchy.....	5-22
Using the Default Directory Hierarchy	5-24
Concept of Active.....	5-26
List Active Catalog.....	5-27
Creating a Part Number	5-29
Mapping Windows	5-31
Mapping with a Logic Symbol	5-32

TABLE OF CONTENTS [Continued]

Lesson 5

Mapping and Catalog Files (Continued)

Adding Power Pins	5-35
Mapping Pins and Setting Swap Codes	5-36
Setting Pin Swap Codes	5-37
Completed Mapping Data	5-38
Mapping Without a Logic Symbol	5-39
Checking Part Number Data	5-42
Creating a Design Catalog	5-43
Lab Exercise	5-44

Lab 5

Mapping and Catalog Files.....5-45

Introduction	5-45
Procedure	5-46
Preparation for Lab	5-46
Reading in Geometries for 74act04 and 74ls373	5-47
Creating a Catalog.....	5-48
Creating a Part Number for 74act04.....	5-48
Defining the Logic Symbol Data	5-49
Mapping Logic Pins to Component Pins	5-51
Creating a Part Number for 74LS373	5-52
Defining the Logic Symbol Data	5-53
Mapping Logic Pins to Component Pins	5-55
Defining Pin Sets	5-56
Checking and Saving Part Numbers and Mapping Files.....	5-57

LIST OF FIGURES

PCB Design Process	1-1
Geometries are Physical Descriptions of Components	1-4
Through-pin Padstack Geometry	1-5
Component Geometry	1-6
Board Geometry	1-7
Examples of Generic Geometry	1-8
Defining a User-defined Layer	1-19
Containers and Design Objects	1-20
Default Directory Hierarchy	1-25
Saving ASCII Geometries	1-28
LIBRARIAN Icon	1-32
Closing the Report Window	1-33
Maximize Window Icon	1-33
Locating the Edit Layer Name	1-34
Completed Logo	1-36
Location of Absolute Coordinate X=0.1, Y=0.1	1-37
Corner of Border	1-37
Completed Border	1-40
Logo with Text	1-42
Completed Logo	1-45
Stroke for Help on Strokes	1-50
View All Stroke	1-51
View Area Stroke	1-51
Card Ejector Geometry	1-52
PCB Design Process	2-1
Set Edit Layer Dialog Box	2-4
View Layers Dialog Box	2-5
Set Grid Dialog Box	2-6
Where to Locate Number of Selected Items	2-8
The Setup Select Filter Dialog Box	2-9
Current Cursor Coordinate Locations	2-10
Origin of the Absolute Coordinate System	2-10
Origin of the Delta Coordinate System	2-11
Lastpoint Icon	2-12
Enter Delta Coordinate Prompt Bar	2-12

LIST OF FIGURES [Continued]

Snap Submenu	2-13
The Add Line Submenu	2-15
Add Line Prompt Bar	2-16
The Outline of a Polygon Defined by Placing a Series of Lines	2-17
The Polygon After Trimming the Ends of the Lines	2-17
The Add Circle Submenu	2-18
The Add Arc Submenu	2-20
The OK to Delete Dialog Box	2-21
Selected a Geometry, and its Basepoint	2-23
Move Prompt Bar and the Location Prompt	2-23
Enter Delta Coordinate Prompt Bar	2-24
Moved Geometry, and its Basepoint	2-25
LIBRARIAN Icon	2-28
The Edit Window Zoom Out Icon	2-30
Experimental Board	2-31
First Four Lines	2-32
Creating a Chamfer at Lower-Left Corner	2-33
The Complete Chamfer	2-33
The First Point of the Slot	2-34
The Second Point of the Slot	2-35
The Roughed-in Slot	2-35
Trimming One Line of the Slot	2-36
Trimming the Second Line of the Slot	2-36
Trimming the Sides of the Slot	2-37
Specifying the Dividing Point	2-38
Making the First Fillet	2-39
The First Fillet	2-39
The Basepoint	2-41
The Circle	2-42
The PCB Design Process	3-1
Geometries Pulldown Menu	3-3
Change This Geometry Popup Menu	3-6
Attributes	3-7
Pin Padstack Attributes	3-9
Via Padstack Attributes	3-11

LIST OF FIGURES [Continued]

Create Component Dialog Box	3-13
Attributes Sub-menu of the Popup Menu	3-15
The Pins Sub-menu of the Popup Menu	3-17
LIBRARIAN Icon	3-20
Completed Resistor Geometry	3-25
Location of the Edit Layer Name	3-28
Resistor Body and Placement Outlines	3-28
Resistor Body Outline	3-29
Location of the Reference Designator	3-31
Resistor Placement Outline Added	3-32
Completed Capacitor Geometry	3-33
The Capacitor Pins	3-34
The Capacitor Body Added	3-35
The Reference Designator Added	3-35
The 96 Pin Connector	3-36
The Pins Array	3-37
The Connector Outline Added	3-39
The Drill Holes Added	3-40
Connector with Placement Outline	3-40
Complete Connector	3-41
The Surface Mount Component	3-42
The Pins of SOIC14	3-45
The Component Outline Graphics Added to SOIC14	3-46
The Placement Outline and Reference Designator on SOIC14	3-47
The Notch added to the SOIC14 Component	3-48
PCB Design Process	4-1
Create Board Dialog Box	4-4
Attributes Popup Menu	4-8
Setup Physical Layers Dialog Box	4-12
Default Physical Layers	4-13
Change Directory Icon	4-16
Outline for Board Geometry	4-18
Beginning of Left End of Board	4-20
Locating the View Out Icon	4-21
Completed Rough Left End	4-23

LIST OF FIGURES [Continued]

Locations for First Chamfer	4-24
Locations for First Fillet	4-25
The Left End Copied to the Right	4-27
The Right Side After Flipping	4-28
The Complete Outline	4-29
Relocating the Origin	4-31
Locating the Basepoint for the First Drill Hole	4-32
Details of Placement Outline	4-35
Details of Routing Outline	4-37
Board Outline with the Left Connector	4-40
Board Outline with Both Connectors	4-41
Final Contents of the Setup Physical Layers Dialog Box	4-44
Completed Via Rules for via040015	4-47
PCB Design Process	5-1
Mapping and Catalog Files	5-3
Example Mapping File for 74ls00	5-5
Pin Mapping for Symbol 1	5-6
Non-Homogeneous Mapping File	5-8
Mapping File Using Common Pins	5-9
Mapping File with Pinsets	5-11
Mapping File with Pinsets for Each Symbol	5-13
74ls244 Mapping File without Pinsets	5-15
A Catalog Directory Structure	5-21
Default Catalog Hierarchy Dialog Box	5-24
Listing of Catalogs with the Active Catalog Indicated	5-26
The Contents of the Active Catalog	5-28
The Create Part Number Dialog Box	5-29
The Edit and Map Windows for Creating a Part Number	5-31
Map Logic Symbol Dialog Box	5-33
The Add Mapping File Power Pins Dialog Box	5-35
The Edit Window used for Pin Mapping	5-36
The Symbol Connections to be Mapped	5-36
The Edit Window with Completed Mapping Data	5-38
The Map Logic Symbol Dialog Box	5-39
The Add Mapping File Pins Dialog Box	5-41

LIST OF TABLES

Top/Bottom Mapping Layers	1-17
Default Library Pathnames.....	1-23
An Example Catalog File	5-17
No Mapping File.....	5-19
An Example of the Default Directory Hierarchy	5-22

About This Training

Welcome to the *Board Station for New Users Training Series*. For information on the tools you learn to use in this training series, see the "About this Training" section of Module 1: *Introduction to Board Station* of the *Board Station for New User's Training Series*.

Workbook Organization

For an overview of the organization and content of all the modules of the *Board Station for New Users Training Series*, refer to section "Workshop Overview" in *Module 1: Introduction to Board Station*.

Related Documentation

For a complete listing of the manuals that make up the PCB documentation set, refer to section "Guide to the Documentation" in the *PCB Products Overview Manual*. The *PCB Products Overview Manual* describes how each manual can help you in the design process. You can find a listing of all Mentor Graphics manuals in the *Mentor Graphics Technical Publications Overview Manual*. Both of these manuals are available in INFORM.

Documentation Conventions

For an explanation of the documentation conventions used in this workbook, refer to the "About this Training" section of Module 1: *Introduction to Board Station* of the *Board Station for New User's Training Series*.

Installation Procedure

For complete instructions on installing the data for this module, refer to "Installation Procedure" in the "About this Training" section of Module 1: *Introduction to Board Station* of the *Board Station for New User's Training Series*.

Lesson 1

Creating Design Geometries

In this lesson, you study the process of developing component libraries and geometries for parts. In Figure 1-1, you can see how this process fits with the overall process of circuit board design.

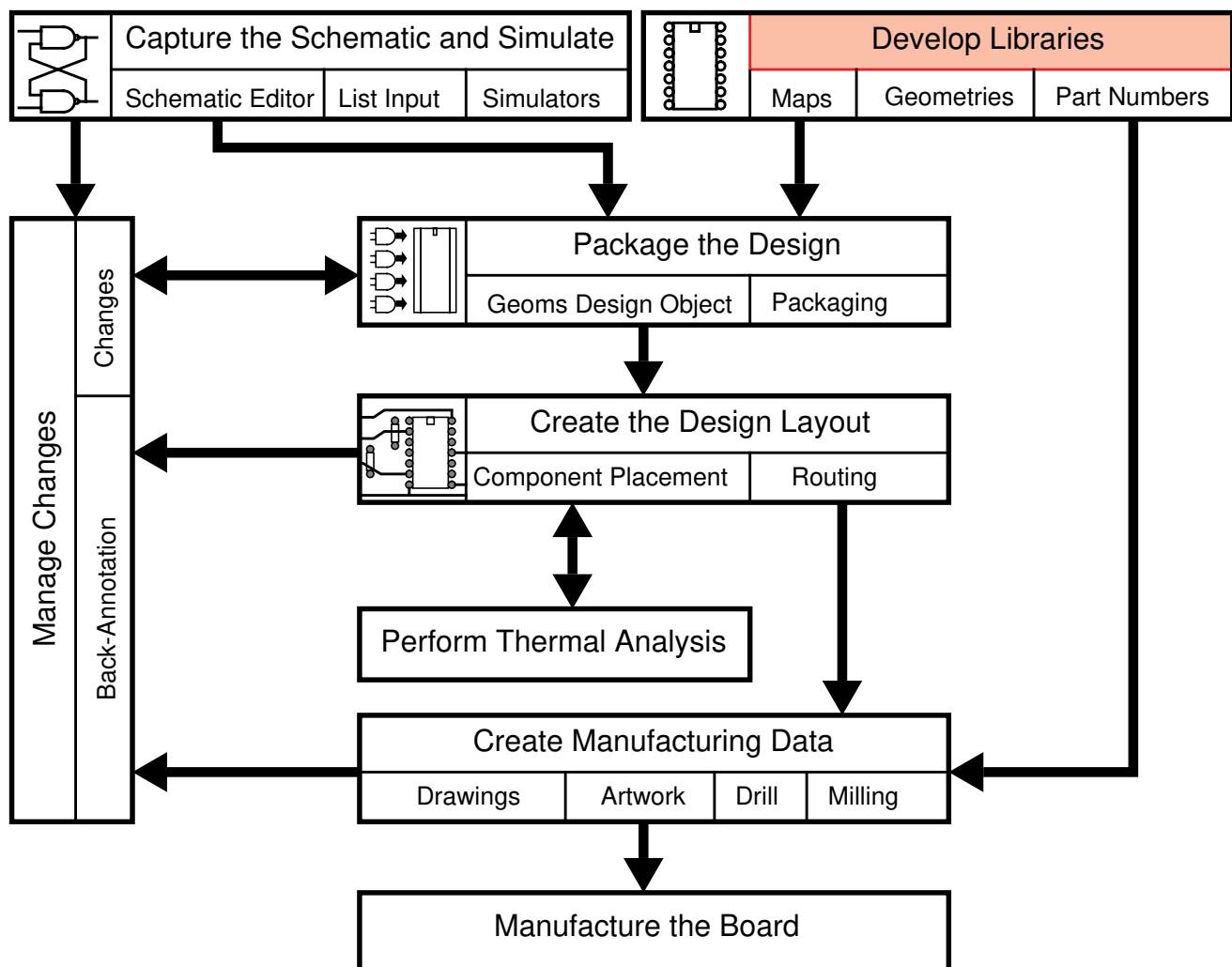


Figure 1-1. PCB Design Process

Objectives

Creating a circuit board design is the process of creating a database of information describing your design. This database contains descriptions of each component on the board, the board itself, and the wiring or trace sizes and locations. Even your design rules are a part of this design database.

The first set of topics in this module describes how to start a printed circuit board design. Then, because geometries define many aspects of the board design, the remainder of the topics describe the structure of geometry libraries and the geometries themselves.

The description of components, the board, and other design features like padstacks comprise a category of data called geometries. These geometries are grouped together in the directories of the file system on your workstation. These directories are called libraries.

As you begin your board design, you determine a source for geometries. You might need to create some geometries yourself. Typically, however, you obtain these geometries from one of the following locations:

- A Mentor Graphics library.
- Your company librarian (the person authorized to create geometries).
- Your company libraries.

Board Station Terminology

The terms for a geometry, part, component, trace, route, and wire are sometimes defined in different ways by different systems. For Board Station tools, these terms are defined as follows:

- **Geometry**—includes the graphics and attribute combinations that describe the physical shape and characteristics of an object used in the board design. The types of geometries include: artwork order geometry, board geometry, component geometry, drawing geometry, generic geometry, panel geometry, and padstack geometry.
- **Part**—refers to a physical device (such as a resistor, capacitor, or IC) used in the board design to perform an electrical function. A part is represented by a component geometry along with a part number for identification, the schematic symbol or symbols packaged in the part, and optional properties.
- **Component**—is used interchangeably with part.
- **Traces**—are the paths that represent the etched or deposited conductor between connected component pins.
- **Route**—is a verb that describes the act of defining the location of traces or wires.
- **Wires**—are pre-formed conductors that are attached to pins to form connections. Examples of wires are the bond wires used in hybrid designs and the insulated conductors used in multiwire designs.

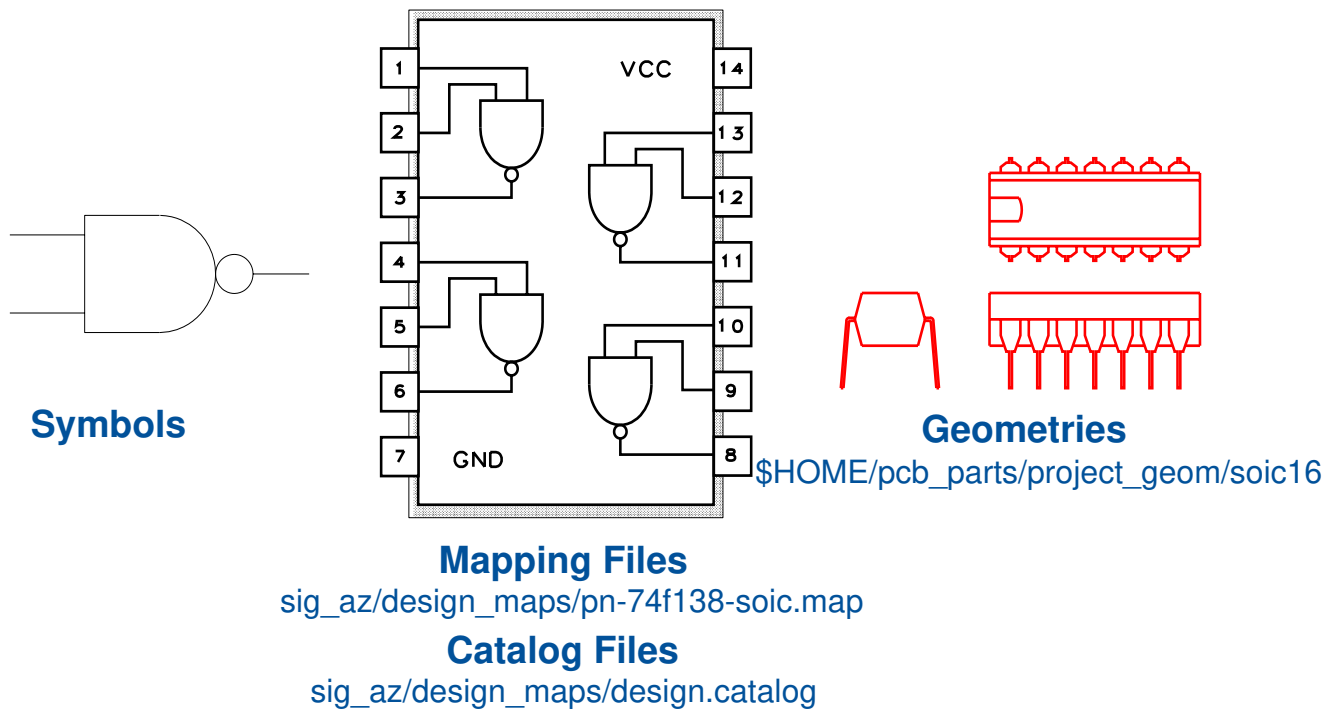


Figure 1-2. Geometries are Physical Descriptions of Components

Basic Geometry Types

Geometries are system representations of electronic and hardware items that are assembled onto a printed circuit board. They are the physical items like resistors, capacitors, SMD packages, and switches. The physical description of the board itself is a geometry. Pin and via padstacks are also geometries. The system also provides for generic geometries like logos and card ejectors.

Geometry types have special meaning in LIBRARIAN. Except for the generic geometry, LIBRARIAN checks for the addition of a set of characteristics, called attributes, to each geometry type.

These are the basic geometry types you use to design a printed circuit board. Other geometry types (such as drawings, panels, and an artwork order) can also be defined. These geometry types are discussed later in the course as a part of the process of creating manufacturing data.

Pin and Via Padstack Geometries

There are two types of padstack geometries: Pin and Via.

- A pin padstack defines the size and shape of the copper land area to which a component pin is attached. A pin padstack can be a through-pin, surface, or blind padstack.
- A via padstack can be a buried via, a 2-layer via, or a simple through-via.

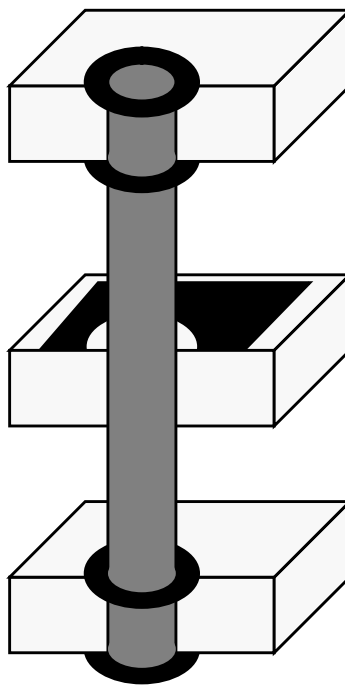


Figure 1-3. Through-pin Padstack Geometry

Component Geometry

A component geometry is a geometry that has one or more pins for electrical connection. These are the electronic items like resistors, SMD components, and switches that are placed on the printed circuit board.

The component geometry defines the location and identification of the pins on the device, the type of pads, and the actual physical size and shape of the geometry. A picture of the device for an assembly drawing and the silkscreen image might also be included.

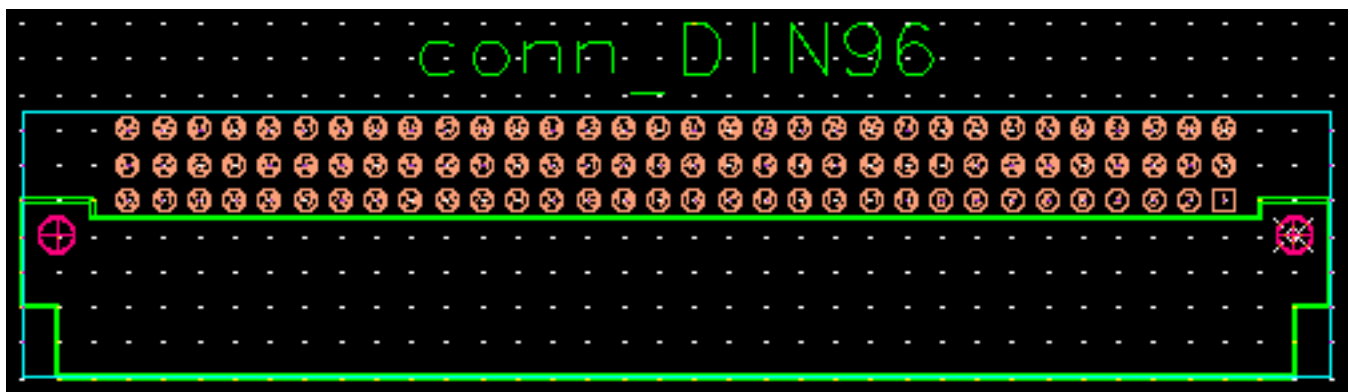


Figure 1-4. Component Geometry

Board Geometry

The board geometry is the definition of the printed circuit board onto which you place components and route traces. The board geometry defines the dimensions of the actual board, the area in which components are allowed, and the area in which routing is allowed.



Figure 1-5. Board Geometry

Generic Geometry

A generic geometry is any picture that is not associated with the physical or electrical characteristics of the board or components. Details for a fabrication drawing are a good example of a generic geometry. Other examples are a logo that is created to identify a board design, alignment targets, and drawing format.

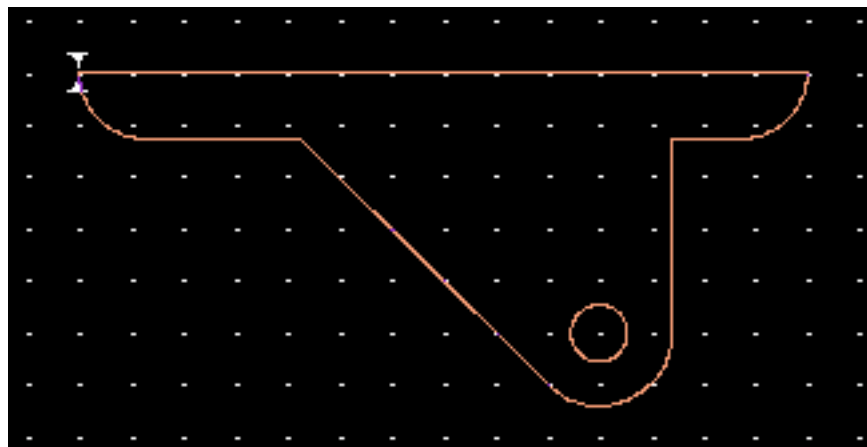
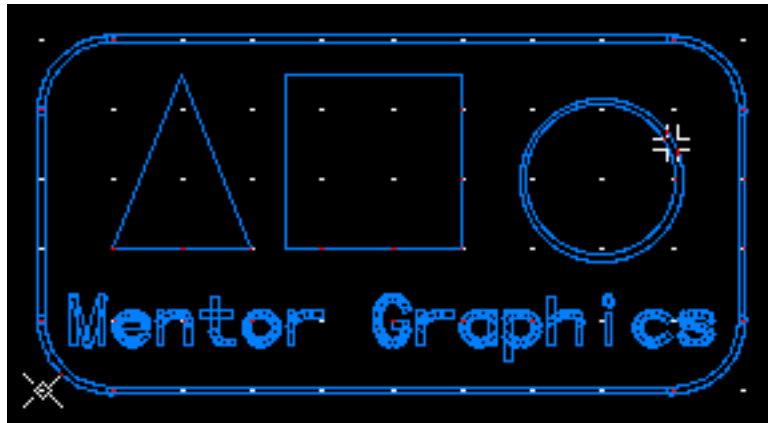


Figure 1-6. Examples of Generic Geometry

Display Layers Categories

In the LIBRARIAN and FabLink tools, you use display layers to control what features of the board you want to view on the display, and also what visible objects are selectable. For example, one display layer shows the board outline, another shows the silkscreen. Using the display layers, you can specify that you want to view the signal layer, but that you cannot edit anything on that display layer unless it is also selectable. There are 1024 logical display layers. Display layers are not the same as the physical board layers.

Layers used for individual printed circuit board designs are defined in the *layers* design object. PCB tools use the Mentor Graphics-supplied default *layer_file* located in *\$MGC_HOME/pkgs/pcb_base/data/layer_file*. The default *layer_file* is first read into the *layers* design object as you begin your design in LIBRARIAN. Layers are saved under the design as *your_design/pcb/layers*.

In the LAYOUT tool, you control the visibility of data by setting the visibility of logical layers and setting the visibility of objects. As a result of the use of objects to display graphical data, LAYOUT uses fewer logical layers than LIBRARIAN and FabLink. Lesson 1 of Module 6 of the *Board Station for New Users Training Series* discusses the use of layers and objects as applied in the LAYOUT tool.

Depending on the type of information stored, the layers in the layer file can be grouped into several functional categories. Display layers are referred to as logical layers in the PCB documentation.

Data reporting—the graphics displayed on these layers are generally system generated and provide you with information to assist you in the printed circuit board design process. The graphics on the Errors layer are automatically generated and display placement and routing design rule violations.

Data storage—the layers in this category store and display user-defined polygon shapes. The shapes generally define areas on the board where special placement, routing, and/or manufacturing rules apply. For example, the Placement_keepout layer displays an area on the board where component placement is not allowed.

Electrical layers—the layers in this category display graphical data related to conductive paths on a printed circuit board and are associated with the physical board layers. For example, SIGNAL_1 generally maps to the first physical routing layer or the top (component) side of the board.

Top and bottom mapping—the layers in this category come in sets of three, consisting of a generic layer and two specific layers. Shapes defined on the generic layer are automatically mapped to the specific layer corresponding to the top or bottom of the printed circuit board.

User-definable layers—you can define special layers to meet your own special design requirements.

Data Reporting Layers

Density, Density_1 to Density_5—the density layer group displays routing predictability data. Graphical data is created on these layers after components are placed on the board and the routability predictor is used.

Errors—the Errors layer displays placement and routing errors on your printed circuit board. After components are placed and routing is started, graphical data is created on this layer when various checking functions are executed.

Off_grid_pins, Off_grid_pins_1, Off grid pins_2—the off_grid_pins layer group displays the locations of all off-grid component pins. Graphical data is created on this layer after the components are placed and the autorouter grid is built in LAYOUT.

Route_grid—the Route_grid layer displays the x, y grid used by the autorouter on all signal layers. Data is created on this layer when you view the autorouter grid or run the autorouter in LAYOUT.

Data Storage Layers

Board_outline—the Board_outline layer displays the board geometry outline. Graphical data is created on this layer whenever you create a board geometry in LIBRARIAN using the userware. You can draw a polygonal shape to represent the board outline.

Dam, Dam_1, Dam_2, Dam_3—the Dam layer set displays polygonal shapes indicating the presence or absence of conductive material on the panel.

Dimension_keepout—the Dimension_keepout layer displays a rectangular area on or around a geometry where horizontal and vertical dimensions are not allowed, if clearance checking is in effect. The system stores the coordinates of the dimension keepout area with the geometry as the Dimension_keepout attribute.

Drill—the Drill layer displays drill symbols that represent through-hole pin and via drill sizes. Graphical data is created on this layer whenever you add the Terminal_drill_size attribute.

Drill_holes—the Drill_holes layer displays drill symbols that represent plated and unplated tooling and mounting holes on a printed circuit board. Data is created on this layer whenever you add the Drill_definition or Drill_definition_unplated attribute in LIBRARIAN or FabLink.

Fixture_outline—the Fixture_outline layer displays the test fixture outline. This polygonal region specifies the boundary of a test fixture geometry. When you create a test fixture with LIBRARIAN, adding the test fixture outline creates the graphical data on this layer and adds the Test_fixture_outline attribute to the geometry.

Milling—the Milling layer displays milling data (plunge and path points, width, arrows) for the board or panel geometry. The shapes on this layer are used to generate milling data.

One-way_region—the One-way_region layer stores the graphics of one-way routing regions. One-way routing regions are rectangular areas that you add to the board or to a component geometry to control the direction of interactive routing within the boundaries of the region.

Panel_outline—the Panel_outline layer stores and displays a path defining the size and shape of the panel geometry. FabLink references the panel outline data during the addition of thieving patterns to the panel.

Placement_keepout—the Placement_keepout layer displays a component keepout area defined for the board. Graphical data is created on this layer whenever you add the Board_placement_keepout attribute and define the keepout area shape in LIBRARIAN or LAYOUT.

Placement_region_1, Placement_region_2—the placement_region layers display an area on the board where a placement region is designated for a particular circuit group, and/or where component height is restricted.

Probe_area—the Probe layers display the probe tip and probe body diameters. Creating a probe geometry with the LIBRARIAN userware adds the graphical data on these layers.

Routing_keepout—the Routing_keepout layer displays an area on the printed circuit board where routing (traces and vias) is not allowed.

Testpoint_1, Testpoint_2—the Testpoint_1 or Testpoint_2 layer displays testpoint symbols indicating identified testpoints. Testpoints probed from the top are identified on the Testpoint_1 layer with the symbol +. Testpoints probed from the bottom are identified on the Testpoint_2 layer with the symbol x. The testpoint symbols appear only for pins and vias that are marked as testpoints for the currently placed test fixture.

Testpoint_keepout—the Testpoint_keepout layer controls the visibility of areas on the board or component geometry in which testpoints are disallowed. The keepout graphical data actually resides on the Pad, Pad_1, or Pad_2 layers. To create the graphical data, add the Testpoint_keepout attribute.

Testpoint_outline—the Testpoint_outline layer displays the testpoint outline. This polygonal region identifies the board region within which testpoints can be mapped or inserted. To create graphical data on the Testpoint_outline layer, add the Testpoint_outline attribute.

Testpoint_reference_1, Testpoint_reference_2—the Testpoint_reference_1 and Testpoint_reference_2 layers display testpoint reference designators. Testpoint reference designators for testpoints probed from the top are stored on the Testpoint_reference_1 layer. Testpoint reference designators for testpoints probed from the bottom are stored on the Testpoint_reference_2 layer.

Trace_keepout—the Trace_keepout layer displays polygonal areas on the printed circuit board where traces are not allowed (although vias are allowed within this area).

Via_keepout—the Via_keepout layer displays an area on the printed circuit board where vias are not allowed.

Via_usage—this layer behaves as a switch to control the viewing of vias. No data is displayed on this layer. When Via_usage is viewed and the Via layer is not, vias are displayed on the corresponding viewed signal layer.

Electrical Layers

Dielectric, Dielectric_n (Hybrid Station)—the dielectric layer set provides areas of nonconductive material between conductors to prevent crossover short circuits. The nonconductive material is called dielectric and provides local insulation at points in a hybrid circuit where a wire bond or another conductor crosses over conductive metal.

Power, Power_1 to Power_8—the Power layers display the power clearance shape for through-hole pin and via padstacks (blind, buried, and segmented vias are also included) for each power layer that does not have its own pad shape defined. The generic Power layer displays the default clearance shape while the specific power layers display the clearance for specific antipad assignments.

Res_bot1 to Res_bot6 (Hybrid Station)—the res_bot layers are resistor layers that occur on the bottom side of the hybrid device. Each res_bot layer is associated with a specific resistive paste (ink). Resistor shapes associated with a specific resistive paste occur on the resistor layer associated with the same resistive paste.

Res_top1 to Res_top6 (Hybrid Station)—the res_top layers are resistor layers that occur on the top side of the hybrid device. Each

res_top layer is associated with a specific resistive paste (ink). Resistor shapes associated with a specific resistive paste occur on the resistor layer associated with the same resistive paste.

Sheet_dielectric, Sheet_dielectric_n (Hybrid Station)—the sheet_dielectric layer set provides layers of screened nonconductive material (dielectric) for insulation between conductor layers on both sides of a multi-layer hybrid device. A sheet_dielectric layer extends to the limits of the hybrid; vias in the dielectric allow connectivity between conductor layers.

Signal, Signal_1 to Signal_12—the generic Signal layer displays the default pad shape of through-hole pin and via padstacks (blind, buried, and segmented vias are also included) used on each routing layer that does not have its own pad shape defined on the specific Signal_n layer. Whatever feature you want to appear on all numbered signal layers you place on the Signal layer. The graphics on the Signal layer are used for creating padstacks. You route traces on the numbered signal layers. The specific Signal_1 to Signal_12 layer displays a pad shape of through-hole pin and via padstacks (blind, buried, and segmented vias are also included) defined for that layer. Graphic data is created on this layer whenever you create the board geometry in LIBRARIAN that uses the specific routing layer and you specify a different pad shape for that specific routing layer. The specific signal layer also displays the routed traces for that layer.

Substrate (Hybrid Station)—the Substrate layer is the base layer for construction of the hybrid circuitry. Layers of a hybrid circuit can occur on one or both sides of the Substrate layer. Your design's physical layer rules must define the position of the Substrate layer. The system understands the relationship between conductor, dielectric, and resistor layers based on the position of the Substrate layer.

Via, Via_n—the generic Via layer displays the default pad shape for via padstacks (blind, buried, and segmented vias are also included) used on each signal layer that does not have its own pad shape defined. The default pad shape is created and resides on the Signal layer. A specific Via_1 to Via_11 layer displays a pad shape for via padstacks (blind, buried, and segmented vias are also included) defined for a specific signal routing layer. The pad shape is created and resides on a specific signal layer (Signal_1 to Signal_12), but to view the via pad shape you must turn on the corresponding via layer.

Top/Bottom Mapping Layers

Breakout—the Breakout layers are used for connecting plating bars. Graphical data is mapped from the generic layer to the specific layer once the components are placed.

Component_body_outline—the Component_body_outline layers display the default component body outline. This polygonal region defines the extent of a component for component-to-probe clearance computations for testpoint generation. In LIBRARIAN you create the component body outline by adding the Component_body_outline attribute to the component geometry.

Drawing—the Drawing layers display text, dimensions, notes, and sheet templates for printed circuit board fabrication and assembly drawings. The specific Drawing_1 or Drawing_2 layer displays the text, dimensions, notes, and sheet templates for fabrication and assembly drawings for the top or bottom sides of the board.

Glue_mask—the Glue_mask layers display the glue application pattern for surface mount components. The graphical data is mapped to the specific layer once the components are placed.

Pad—the Pad layers display the pad shapes used for the surface mount components placed on the top and bottom sides of the board. The pad shape is created on the Pad layer whenever a surface mount pad geometry is created and the same pad shape for both sides is requested. The pad shape is created on the specific Pad_1 or Pad_2 layer whenever a surface mount pad geometry is created and a different pad shape for the top and/or bottom side of the board is requested.

In LAYOUT, pad layers do not exist. Shapes on the pad layers display on the corresponding signal layers. For example, a shape drawn on layer Pad_1 displays on layer Signal_1 if you also select to view the Pads object.

Paste_mask—the Paste_mask layers display the default solder paste pattern used for surface mount components. The data is mapped from the generic layer to the specific layer once the components are placed.

Pin_id—the Pin_id layer displays the pin padstack locations and defines routing terminations on components. Graphical data is created on the generic Pin_id layer whenever you add pins while building a component. The specific Pin_id layers display the pin padstack locations on the components placed on the top or bottom side of the board.

Place—the Place layers display the placement outlines of components. Data is automatically mapped to the Place_1 or Place_2 layers once the components are placed in LAYOUT.

Probe—the generic Probe layers display the default probe tip and probe body diameters. Creating a probe geometry with the LIBRARIAN userware adds the graphical data on this layer.

Probe_symbol—the Probe_symbol layers display the user-defined graphic symbol for the probe. Creation of a probe symbol is an optional step in creating a probe geometry. The generic Probe_symbol layer is available only in LIBRARIAN. In LAYOUT, the specific Probe_symbol_1 layer displays the user-defined graphic symbols of probes used to probe testpoints on the top side of the board. The specific Probe_symbol_2 layer displays the user-defined graphic symbols of probes used to probe testpoints on the bottom side of the board.

Silkscreen—the Silkscreen layers display the silkscreen text and shapes of components. The graphical data is mapped to the specific layer depending on the top or bottom placement of the component.

Solder_mask—the Solder_mask layers display the default soldermask shapes of pads for components used on both the top and bottom side of the board. Graphical data is placed on a specific layer whenever you create a pad geometry in LIBRARIAN and you specify a different shape for the top or bottom side of the board.

Thermal—the Thermal layers display thermal analysis information. You can view these layers in LAYOUT only. Graphical data is created on this layer via an attribute in AutoTherm. The specific Thermal_1 or Thermal_2 layer displays thermal analysis information for the top or bottom side of the board.

Trim_path (Hybrid Station)—the Trim_path layers display the graphics of a path on an ink resistor geometry. The coordinates of the path define a channel for trimming the resistor value during manufacture. In LIBRARIAN or FabLink, add the trim path to the Trim_path layer when the trim path applies to both top and bottom placement of the resistor. In LIBRARIAN or FabLink, add the trim path to the Trim_path_1 layer when the trim path applies to top placement of the resistor. Add the trim path to the Trim_path_2 layer when the trim path applies to bottom placement of the resistor.

Table 1-1. Top/Bottom Mapping Layers

Generic Layer	Top Layer	Bottom Layer (always _2 regardless of number of board layers)
Breakout	Breakout_1	Breakout_2
Drawing	Drawing_1	Drawing_2
Glue_mask	Glue_mask_1	Glue_mask_2
Pad	Pad_1	Pad_2
Paste_mask	Paste_mask_1	Paste_mask_2
Pin_id	Pin_id_1	Pin_id_2
Place	Place_1	Place_2
Silkscreen	Silkscreen_1	Silkscreen_2
Solder_mask	Solder_mask_1	Solder_mask_2
Thermal	Thermal_1	Thermal_2

User-Defined Layers

When you invoke LIBRARIAN, LAYOUT, or FabLink on a design, the layer definitions are read in from the *layers* design object in the *pcb* container if the design object exists, or from the default layer file *\$MGC_HOME/pkgs/pcb_base/data/layer_file*. You can modify the *layers* design object for a design to suit your needs. Your system manager can modify *\$MGC_HOME/pkgs/pcb_base/data/layer_file* to suit the needs of your entire worksite.

For example, the default *layer_file* file defines 12 signal layers and 8 power layers. If you need more signal and power layers for a particular design, you must add the layers to the *layers* design object for that design.

Use these guidelines if you modify the *layers* design object or if your system manager modifies the default layer file.

- Only add new layers. Deleting unused layers is not necessary.
- Use new layer names for the layers you create.
- Layer sets use consecutive stacking numbers. If the new layer you create is an addition to a layer set, assign the next consecutive number.
- When adding a single layer, use a stacking number that is not assigned to another layer and use numbers in the ranges 200 to 255 and 800 to 1023, except do not assign 201, 202, 203, 237, 238, 241, and 242.
- When adding a new layer, be sure to define all parameters.

For example, you want to be able to draw your company logo on a special layer. You define a layer named Logo and use the unused stacking number 204.

Select menu item **Setup Design Rules > Logical Layers**. You fill out the Setup Logical Layers dialog box in LIBRARIAN, as shown in Figure 1-7.

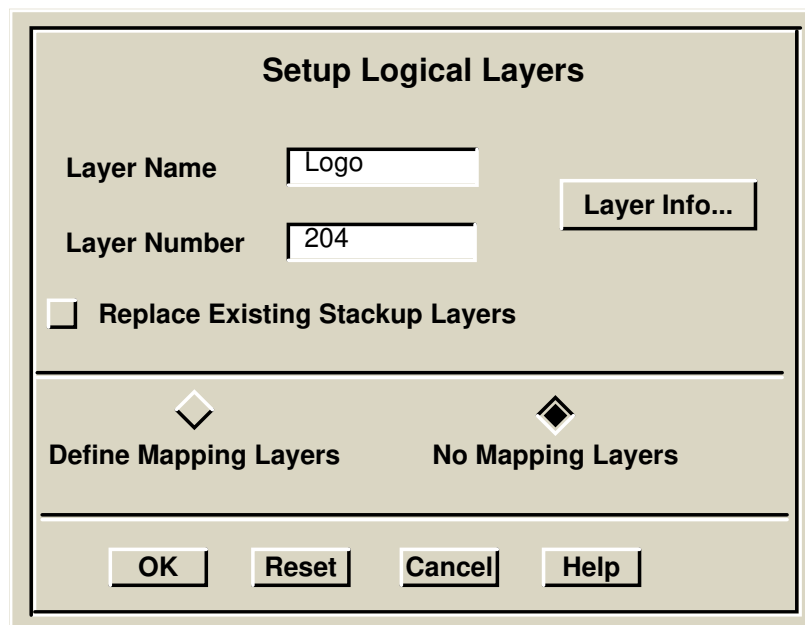


Figure 1-7. Defining a User-defined Layer

For more information on the process of modifying the *layers* design object, refer to section "Creating Customized Layers" in the *PCB Products Design Reference Manual*.

Containers and Design Objects

The file system on your workstation contains directories and files. The printed circuit board design tools interpret directories and files as containers and design objects.

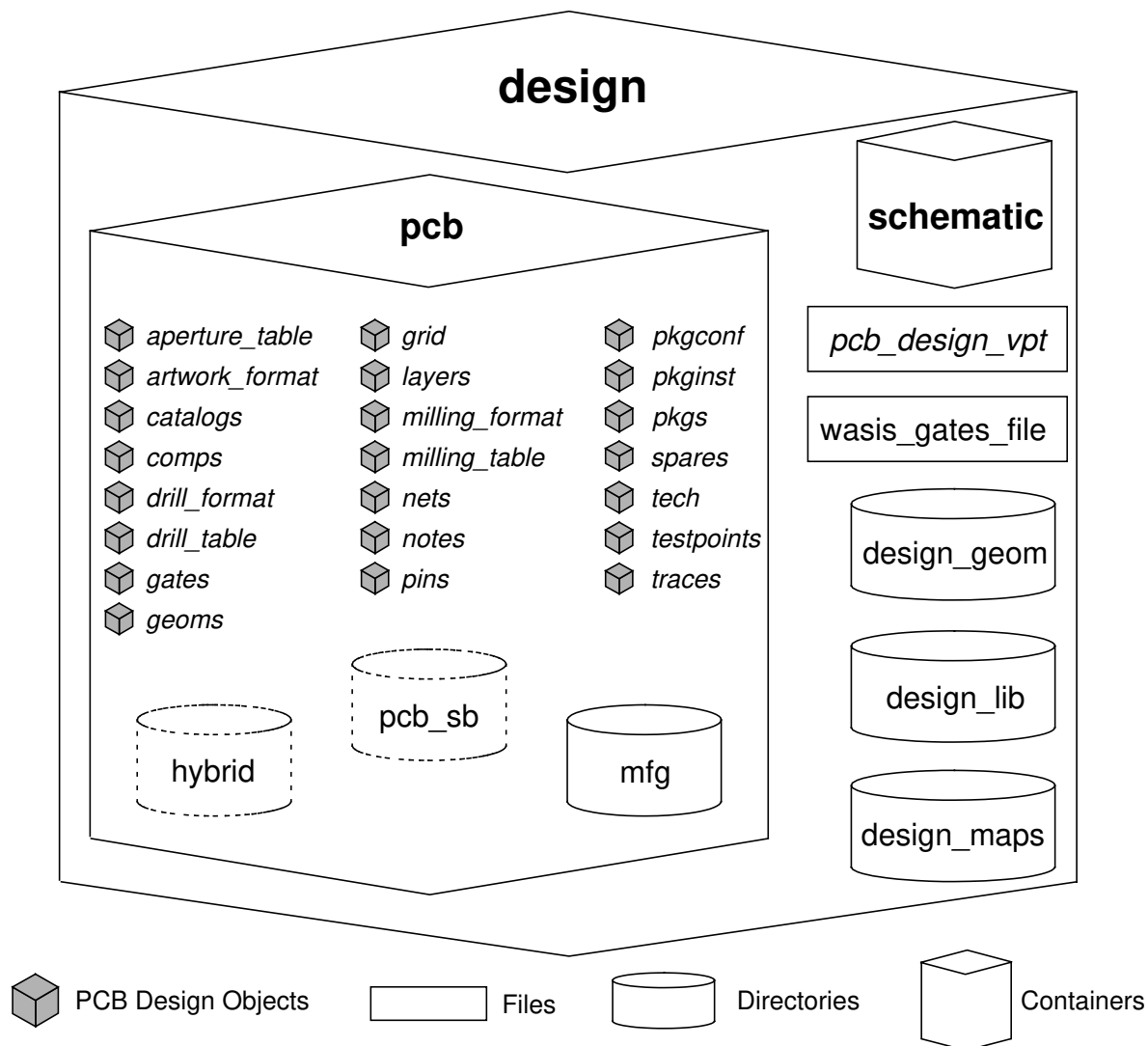


Figure 1-8. PCB Design Data Organization

- Containers are file system directories with supporting attribute files.
- LIBRARIAN creates the contents of a PCB design object, which includes a *pcb* container to hold all PCB design data and output files. The directory structure allows the PCB design object to be treated as a single design database.

- The *mfg* container, created within the *pcb* container, contains all outputs such as artwork files and various reports.
- The *design_geom*, *design_maps*, and *design_lib* directories are a part of the PCB design object.

Library Management

The LIBRARIAN tool provides tools, checking capability and library directory structure to facilitate geometry creation and maintenance. Good library management is a system involving people, processes, and tools. The library management system affects the entire product development and manufacturing process. You must define a system that protects the integrity of complete and checked parts. A company librarian or a circuit designer may also need to create unique parts for a particular design. The LIBRARIAN directory hierarchy allows you to define which directories are used for particular purposes. You can tailor the library management system to suit the needs of your site.

Library Management System

In addition to using the LIBRARIAN tool, you can use the Library Management System. The Library Management System (LMS) creates your libraries in a consistent fashion to meet all requirements for all phases of the board design process at your site. LMS is a Mentor Graphics product that provides a structure and supporting utilities for the creation, storage, retrieval, and use of PCB-oriented parts in a user production environment. LMS libraries then are the libraries that a site creates and maintains using LMS tools. LMS offers three distinct environments within which you can create your site's libraries:

- *Source libraries* serve as repositories for information that includes Mentor Graphics libraries, in-house libraries, libraries licensed from third parties, unqualified parts for use as sources for further model development, or LMS libraries from other divisions, projects, or libraries.
- *Development libraries* provide workspace for librarians to create, modify, assemble, test, and approve parts. Typically, when a part meets company requirements, the librarian then transfers it from a development library to a release library for general use.

- *Release libraries* contain the parts that have been fully qualified for use in final designs. To protect the integrity of qualified parts, LMS does not allow editing of part data in release libraries.

For more information on LMS and related LMS libraries, refer to the *Library Management System User's and Reference Manual*.

Default Directory Hierarchy

Mentor Graphics creates a default hierarchy for library directories. Because companies design, build, and use libraries in a variety of ways, separate types of libraries are supported to meet the needs of each company.

- **Mentor Graphics Libraries**—contain an extensive set of symbols, catalogs, and geometries. These libraries form an excellent starter library, but the data contained typically needs some modification to meet the standards of each company.
- **Company Libraries**—can contain symbols, catalogs, and geometries that meet specific company standards and have been approved for use on printed circuit board designs.
- **Project Libraries**—can contain symbols, catalogs, and geometries that meet standards beyond those used in the design of the company libraries. These libraries allow you to work with multi-standard projects.
- **Design Libraries**—can contain symbols, catalogs, and geometries that were designed to meet the requirements of a specific design. Because the design libraries are contained in the design container, they are part of the design database. Your training design data includes a design library that you use as you work through the labs.
- **User Libraries**—can contain symbols, catalogs, and geometries that need temporary storage or that have not been company approved.
- **Other Libraries**—can contain symbols, catalogs, and geometries that are not part of any of the above libraries, but must be accessible to designers for the design process.

Table 1-2. Default Library Pathnames

Directory Level	Default Pathname
Mentor	\$MGC_PCBPARTS/pcb_geoms
	\$MGC_PCBPARTS/pcb_maps
	\$MGC_PCBPARTS/pcb_libs
Company (optional)	\$HOME/pcb_parts/company_geom
	\$HOME/pcb_parts/company_maps
	\$HOME/pcb_parts/company_lib
Project (optional)	\$HOME/pcb_parts/project_geom
	\$HOME/pcb_parts/project_maps
	\$HOME/pcb_parts/project_lib
User	\$HOME/pcb_parts/user_geom
	\$HOME/pcb_parts/user_maps
	\$HOME/pcb_parts/user_lib
Design	design_pathname/design_geom
	design_pathname/design_maps
	design_pathname/design_lib
Other	any_pathname

Geometry files are found at each level of the hierarchy in the directory named:

`<level_name>_geom`

The name of the geometry directory is slightly different at the Mentor Graphics level of the hierarchy. At the Mentor Graphics level, the `<level_name>` becomes `pcb`, as follows:

`<level_name>_geoms` is `pcb_geoms`

Geometry files provided by Mentor Graphics are located in the directories under the *\$MGC_PCBPARTS/pcb_geoms* directory.

Depending on how your system manager sets up your system, these directories (and links) are automatically created when you invoke LIBRARIAN. LIBRARIAN creates the *user_geom* directory in the *\$HOME/pcb_parts* directory. LIBRARIAN also creates the directory *design_geom* in your design directory if the directory does not exist when you invoke LIBRARIAN on a design. Optionally, LIBRARIAN creates directories or links to directories at the Company and Project levels.

The directory level called Other is for users who do not want to use the default directory hierarchy, or who have existing directories and files outside of the default directory hierarchy. The Other directory does not have a predefined location, rather it is simply a category that allows you to enter an absolute pathname to access data outside the directory hierarchy. LIBRARIAN stores the pathnames to catalogs accessed at the Other directory level (if the pathnames have been read into the LIBRARIAN session) and restores the pathnames at the next LIBRARIAN session. LIBRARIAN writes the pathnames in the file *\$HOME/pcb_parts/other_pathnames*.

Using the Default Directory Hierarchy

Using the default directory hierarchy, you can read and write files in the libraries without knowing the absolute pathnames to the files. You can view the default geometry library directory hierarchy at any time by choosing the menu item:

Geometries > List Geometry Libraries...

A dialog box opens to show the levels in the default directory hierarchy. The dialog box dims the level name of any level that has no geometries. The levels that currently contain geometries are available. Available levels can differ from session to session, and from user to user, because the levels represent directories that can be modified (except the Mentor level). Along with the default directory level names, the dialog box also displays the pathnames for each level and reports the number of geometry libraries at each level.

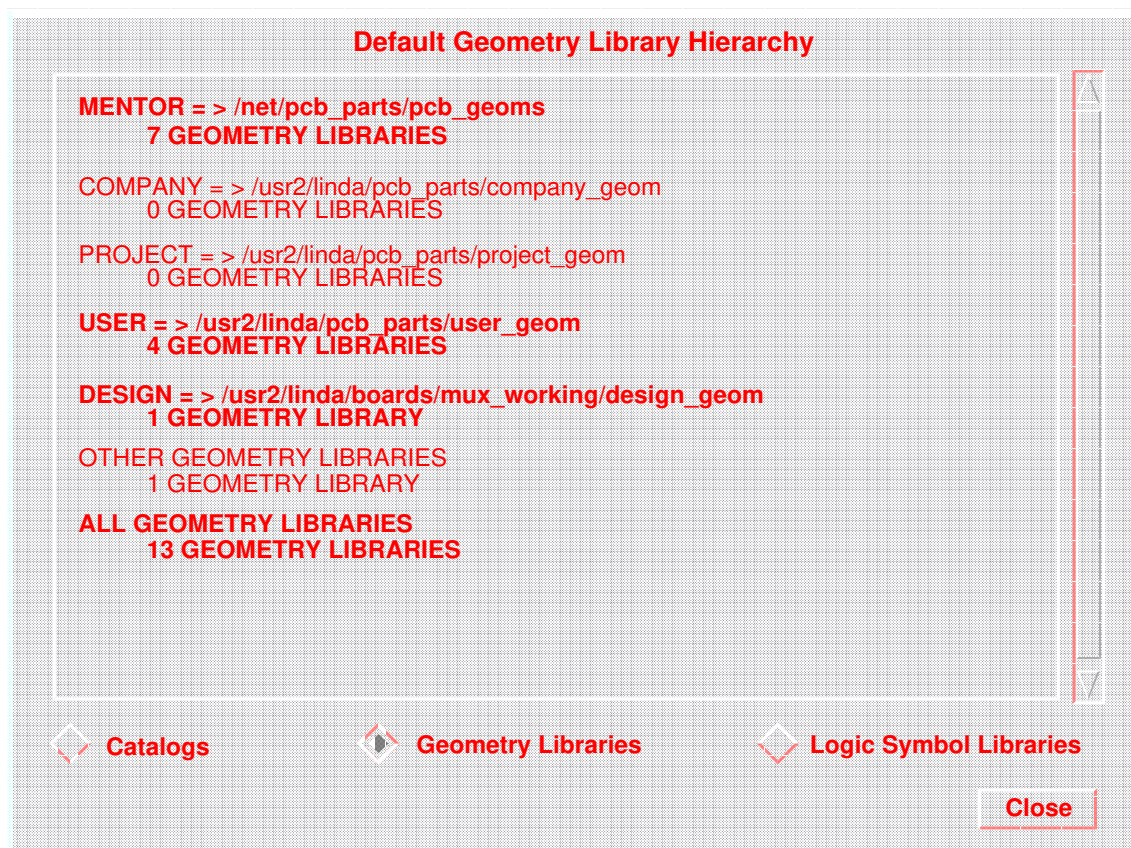


Figure 1-9. Default Directory Hierarchy

To see a listing of the geometry libraries at one of the levels, you select the level by placing the cursor over the level name and clicking the Select mouse button.

The dialog box changes to show a listing of the geometry libraries at the selected level. Read and View options in the dialog box provide a way for you to read or view the library contents. The listing indicates whether a geometry library has been read into the LIBRARIAN session. The listing also shows whether one of the geometry libraries at the level is the active geometry library.

You can select any or all of the geometry libraries in the listing. Pressing the Read button at the bottom of the dialog box reads a selected geometry library into the LIBRARIAN session. Pressing the View button lists the contents of a selected geometry library.

From the listing of the library contents, you can select and read one or more items in the listing.

Libraries

Three different types of library information are associated with components used in the design of printed circuit boards: Symbol Libraries, Catalog Libraries, and Geometry Libraries.

- **Symbol Libraries**—contain groups of schematic symbols. Symbols are created with a symbol editing tool. The symbols are found on schematic drawings showing the function of a circuit. Mentor Graphics symbols contain information about the electrical characteristics of the symbol's function. Symbols can also contain part number, pin number, pin name information, and, optionally, other properties.
- **Catalog Libraries**—contain mapping files and a catalog file. A mapping file describes the relationship of symbols to a physical component. The catalog file is a database of part numbers. Each part number in the catalog file is described by a symbol, mapping file, and geometry name.
- **Geometry Libraries**—contain a description of the physical aspects of a part. This includes the location of pins, the number of the pins, and the size and shape of the physical part, and reference designator text.

Saving Geometries

How you save your geometries depends on whether or not you invoked LIBRARIAN on a design or stand-alone, and in what form you want your geometries saved.

In general, you can either save geometries with a design, or you can save them as ASCII files in a library. If you save geometries with a design, they are saved as binary format files in a *pcb* container under the design. If you save geometries in ASCII files in a library, they are not associated with a design, and they can be read into any design.

If you invoke LIBRARIAN on a design, you can either save geometries with a design, or you can save geometries to ASCII files in a library.

If you invoke LIBRARIAN in stand-alone mode, you can only save geometries to ASCII files in a library.

If you invoked LIBRARIAN on a design, you can save the design, including all the geometries in the design, using the **File > Save > Design All** menu item. If you save your design this way, it is saved in a series of files in the *pcb* container (directory) under your design (component) container. For example, the design you will work with later in this training is called *sig_az*. If you invoked LIBRARIAN on the *sig_az* design, and then later saved the design, the design would be saved in files under the container: *your_path/sig_az/pcb*. When you save the design, the geometries are saved in binary format in a design object named *geoms*.

If you want to save your geometries in ASCII format and not have them associated with any design, choose the **File > Save > Save ASCII Geometries** pulldown menu item.

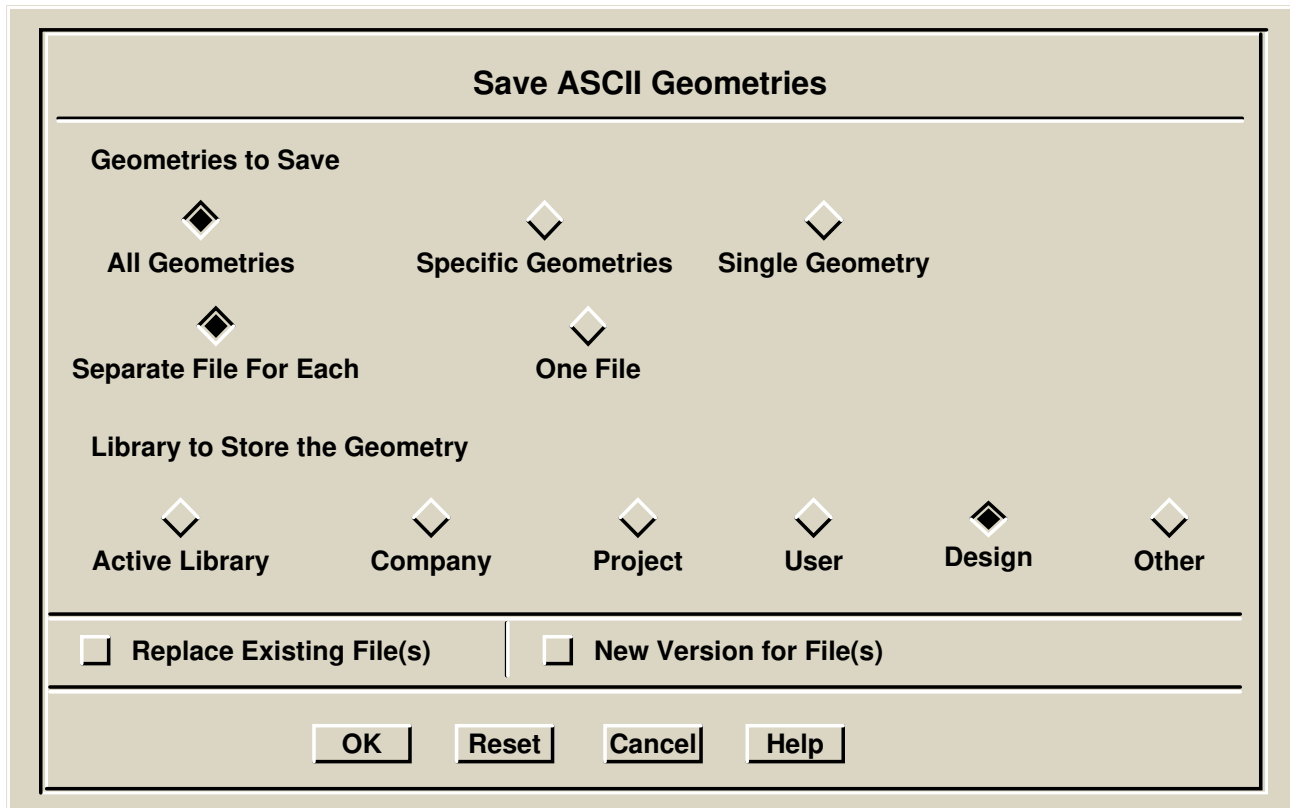


Figure 1-10. Saving ASCII Geometries

The Save ASCII Geometries dialog box opens with the following choices:

- **Geometries to Save**—at this prompt, choose one of the following responses to indicate which geometries to save:
 - **All Geometries**—press this button to save all the geometries in the current session. With this response, you have the choice of saving each geometry in a separate file or saving all geometries in one file. The dialog box displays two buttons for entering your choice.
 - **Specific Geometries**—press this button to save one or more of the geometries in the current session. A list box labeled Geometry Names appears and displays the names of all geometries in the current session. Select the geometries to save.

- **Single Geometry**—press this button to save one geometry from the current session. A list box labeled Geometry Name appears and displays the names of all geometries in the current session. Select the geometry to save. In the File Pathname entry box, enter the pathname for the file.
- **Library to Store the Geometry**—press one of the following buttons to direct the file or files to the Active Library, Company, Project, User, Design, or Other library.
- **Replace Existing File(s)**—press this button to overwrite an existing version of a geometry file with the file or files you are saving.
- **New Version for File(s)**—press this button to create a new version of a geometry file or files while retaining an existing version.

Lab Exercise

This lab exercise familiarizes you with the some of the basic commands used in Board Station for creating and manipulating graphics. You learn how to change the setup parameters and you become familiar with the library structure and how to save parts.

Upon completion of this lab exercise you can:

- Create simple graphic elements.
- Read existing geometries into LIBRARIAN.
- Modify geometry graphics.
- Save geometries to a user library.

Turn to Module 3—Lab 1: "Introduction to LIBRARIAN".

Lab 1

Introduction to LIBRARIAN

Introduction

This lab exercise introduces you to some basic commands used in LIBRARIAN for creating and manipulating graphics. You will learn how to change setup parameters. You will also become familiar with the library structure and how to save parts.

Upon completion of this lab exercise you will be able to:

- Create simple graphic geometries.
- Read existing geometries into LIBRARIAN.
- Modify the graphics of an existing geometry.
- Save geometries to a parts library.

Procedure

In this procedure, you invoke LIBRARIAN and create a logo and card ejector.

Preparation for Lab

In this procedure, you use the LIBRARIAN tool to create geometries for your design.

1. If you or your instructor have not already done so, complete the Installation Procedure in the About This Training section of this manual.

2. Invoke the Design Manager by entering the following in a shell:

```
$MGC_HOME/bin/dmgr
```

3. Find the LIBRARIAN icon in the Tools window, as shown in Figure 1-11. Invoke LIBRARIAN by placing the cursor on the LIBRARIAN icon and double clicking the Select mouse button.



Figure 1-11. LIBRARIAN Icon

The cursor changes from an arrow to an hour-glass shape indicating that LIBRARIAN is invoking. If the cursor does not change shape, double-click again on the LIBRARIAN icon. You must double-click quickly.

4. In the Specify Invocation Mode dialog box that appears, choose **Invocation Mode: Stand Alone**. Then select the **OK** button in the dialog box.

A Report-Startup message might appear in the middle of the LIBRARIAN Session window. This report is a list of notes concerning the files used to invoke the LIBRARIAN tool.

5. After reading the report notes, close the report window as shown in Figure 1-12, and then maximize the size of the LIBRARIAN session window to fill the display by clicking the Select mouse button on the Maximize window icon shown in Figure 1-13.

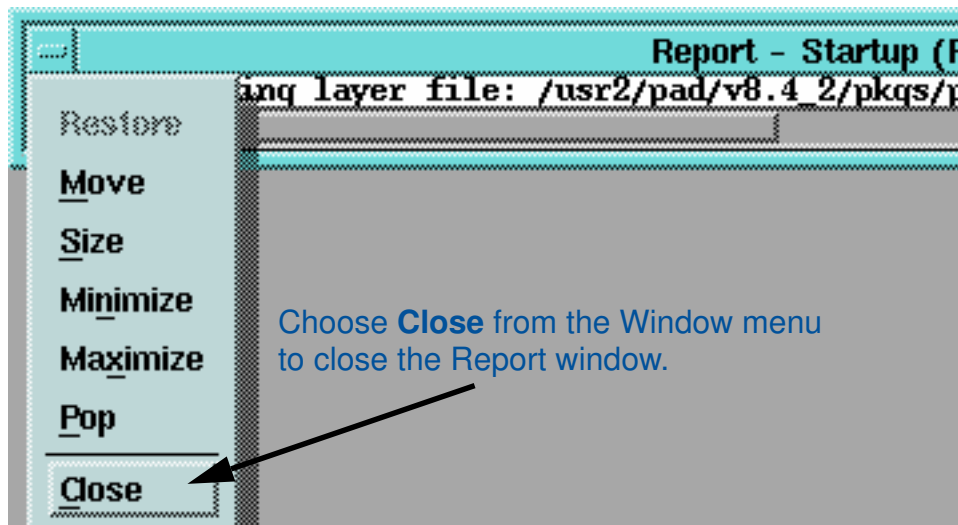


Figure 1-12. Closing the Report Window

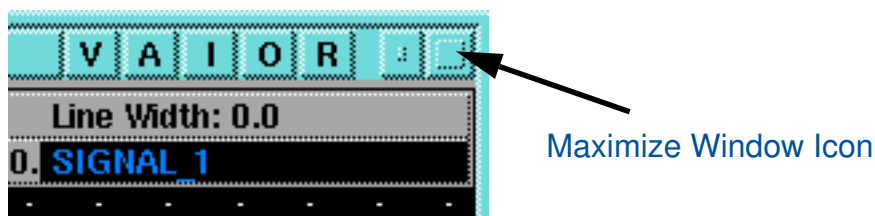


Figure 1-13. Maximize Window Icon

Setting Up to Create the Geometry Logo

The first geometry you build is a logo. Even though the logo is non-electrical, the techniques you use to construct the logo geometry are identical for all types of geometries. Later, you will learn more about creating geometries, but the basic techniques you learn here will help you understand more complex techniques.

1. Open an Edit window for a new geometry and set the edit layer by choosing the **Geometries > Create Geometry > Generic...** menu item.
2. When the dialog box appears, enter **logo** as the geometry name. Because the logo is a non-electrical graphic part, the name used is not critical.

Leave the remaining options in the dialog box as they are.

Complete the dialog box by either pressing the **OK** button at the bottom of the dialog box or by pressing the RETURN key on the keyboard. You can execute almost any dialog box or prompt bar using either the RETURN key or the OK button.

3. Check the upper-right corner of the Edit window, as shown in Figure 1-14, and verify that the current edit layer is **SIGNAL_1**.

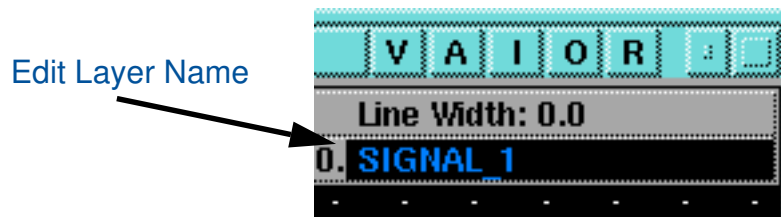


Figure 1-14. Locating the Edit Layer Name

To etch the logo in copper as part of the component side of the board, the logo graphics must be placed on the **SIGNAL_1** layer. If the edit layer is set to something other than **SIGNAL_1**, press the Set Edit Layer function key and select **SIGNAL_1** from the dialog box by clicking the Select mouse button on the **SIGNAL_1** line in the dialog box. Then **OK** the dialog box.

4. Set up the display grid by choosing the **Setup > Grid...** menu item.
5. Set the **X Increment** to **.05**. Leave the **Y Increment** box empty.

If you enter only the X value, the system uses the same value for both directions.

6. Leave the **X Offset** and **Y Offset** blank. Enter a **2** for the **Display Interval**. Finally, **OK** the dialog box.

A display interval of 2 causes the cursor to snap to the 0.05 grid but the screen displays a 0.1 grid making it easier to measure and locate specific locations.

7. Set up the line width by choosing the **Setup > Line Width:** menu item. Enter **.01** and **OK** the prompt bar.

Later, when you generate the artwork data, the system selects the proper aperture size to draw the border of your logo. The line width setting is visible in the upper-right corner of the Edit window.

Creating the Logo Border

Now you create the outline of the logo shown in Figure 1-15.

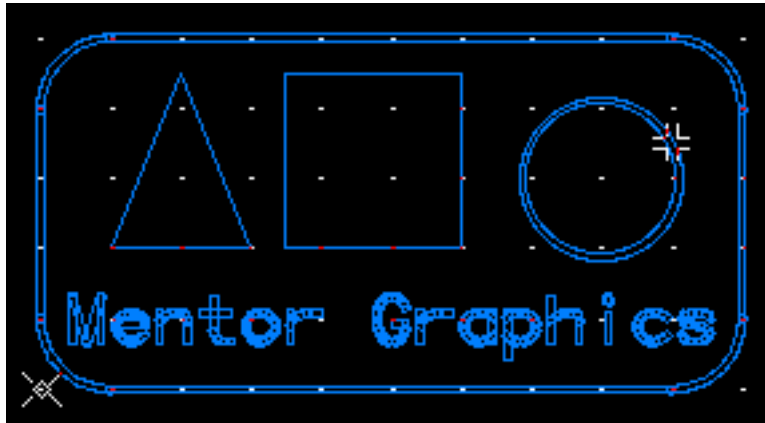


Figure 1-15. Completed Logo

First, you create the arcs for the corners of the logo border. The logo border is a box with rounded corners. The size of the logo is 1.0 inches wide by 0.6 inches high.

As you work through this lab, consider that the purpose of this exercise is to help you become familiar with LIBRARIAN, the user interface, and some of the geometry creation functions. The procedure you use to create the logo is not the most efficient method of creating this geometry. However, this procedure is used because it shows many geometry creation techniques with a simple geometry. In later labs, you will concentrate on learning more efficient procedures.

1. Choose the [Top Menu] **Shapes > Add Arc > Radius Angles:** popup menu item from the edit window. In the prompt bar, enter **180** at the Start Angle prompt. Press the TAB key to highlight the Degrees of Arc prompt, and enter **90**. Press the TAB key again to highlight the Radius prompt, and enter **1**. Press the TAB key to highlight the location prompt.

The cursor becomes a full screen cursor indicating that you are to locate a center point for the arc. You will specify the center point in the next step.

2. Place the cursor at the first grid point to the right of the geometry origin (where the basepoint is at approximately the center of the edit window) and one grid up, as shown in Figure 1-16. To verify you have the correct location, look at the Absolute coordinate location (Abs) displayed in the status window. The Absolute coordinate location must be 0.1, 0.1. When the cursor is correctly located, click the Select mouse button.

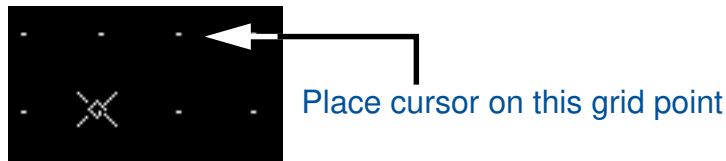


Figure 1-16. Location of Absolute Coordinate X=0.1, Y=0.1

When you click the Select mouse button, the arc is created as shown in Figure 1-17.



Figure 1-17. Corner of Border

From now on, when you see a menu path that starts with a menu name in brackets, such as **[Edit]**, it means the menu is a popup found in an edit window. Because the menu contents and name can change, depending on what is selected, the name in brackets changes to reflect the name of the menu in the edit window during that operation.

Also, now that you have learned to use the TAB key to move between prompts in a prompt bar, only the values of the prompts are provided for you. You will not receive instructions for entering them into the prompts. If you repeatedly press the TAB key, the highlight cycles through all the prompts, and then repeats from the beginning of the prompt bar. If you press the SHIFT-TAB key, the highlight moves from prompt to prompt in reverse order.

3. Press the **Cancel** button in the Add Arc Radius Angle prompt bar before continuing.

Most graphics functions automatically repeat by default. This setting enables you to add several graphic elements without again choosing the function from the popup menu.

4. Choose the **Setup > Display Environment** menu item. In the dialog box that is displayed, choose Show Scrolls. **OK** the dialog box.

Scroll bars are now visible in the edit window. Use the scroll bars and arrows to scroll and position the image as needed.

5. Place the cursor on the right-pointing scroll arrow at the base of the edit window, and click the Select mouse button. Keep clicking on the scroll arrow until the arc is in the left half of the view.

6. Choose the **[Top Menu] Shapes > Add Arc > Point Angles:** menu item.

The prompt bar already has the 3 points prompt highlighted.

7. Position the cursor at absolute x,y location (0.9,0.1) as shown in the status window, and click the Select mouse button. Move the cursor down one displayed grid (to 0.9, 0.0) and click the Select mouse button again. A dynamic circle is displayed. Move the cursor up and to the right one displayed grid (to 1.0, 0.1) until only the lower-right corner of the circle is displayed. Click the Select mouse button. Press the **Cancel** button in the Add Arc Point Angle prompt bar before continuing.

The second arc is now in place. Next, you will select and copy the lower two arcs to create the upper two arcs of the logo outline.

8. Place the cursor on the first arc and click the Select mouse button. The arc turns white indicating that it has been selected. Move the cursor to the second arc and do the same thing.

With both arcs selected, you will copy them in the next step.

9. Choose the **[Top Menu] Shapes > Copy > Copy:** menu item.

10. Move the cursor (and the ghost image of the arcs) up 0.5 inches (5 displayed grids) to the absolute coordinate 0.9, 0.5. Click the Select mouse button to copy the arcs. Cancel the copy prompt bar.

There are now copies of the original arcs. You will now flip them to the correct orientation.

11. Make sure the copies of the arcs are still selected. Choose the **[Top Menu] Shapes > Flip > Vertically** menu item.

The arcs are now in the correct position to be corners for your border. They are still selected.

12. If a prompt bar reappears, cancel it, then unselect all items by either choosing the **[Top Menu] Shapes > Unselect** menu item, or by pressing the Unselect All function key.

You might find that the Unselect All function key is much more handy to use. Both the menu item and the function key do exactly the same thing. You will need this function frequently.

Next, you will add the lines between the arcs.

13. Choose the **[Top Menu] Shapes > Add Line > Add Line:** menu item.

The prompt bar appears with the location prompt highlighted.

14. Add a line by placing the cursor at the end of an arc, clicking the Select mouse button, moving the cursor to the end of the next arc, and again clicking the Select mouse button. To complete the line, **OK** the prompt or press the RETURN key.

The line is completed. The Add Line prompt bar returns, because the repeat option is on.

15. Complete the remaining lines of the border in the same manner.

To remove the Add Line: prompt, press the Cancel button on the prompt bar. Your completed border looks similar to **Figure 1-**

18. Completed Border

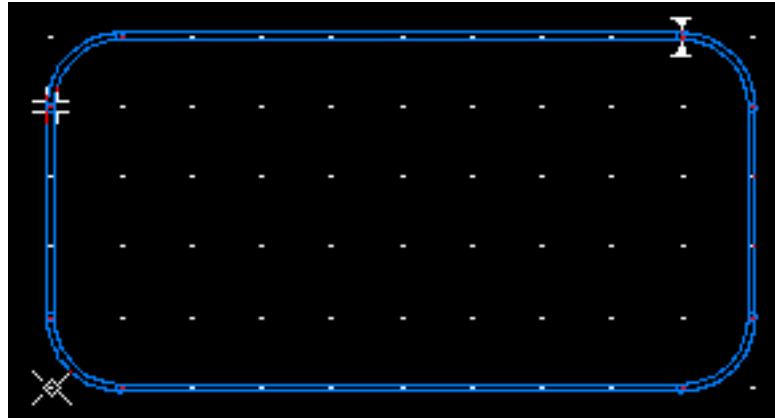


Figure 1-18. Completed Border

Creating the Logo Text

Now you add the text to the logo. Before adding text, check the defaults and set the parameters.

1. Choose the **Setup > Text...** menu item.
2. Change the following entries in the dialog box, then **OK** the dialog box:

Height: **.062** Justification: **Center Center**

This sets the text parameters for any text you add to the geometry.

3. Add text by choosing the **[Top Menu] Text > Add Interactive Text:** menu item. Next, position the cursor at absolute coordinate X=0.5, Y=0.1 and click the Select mouse button.

A text extents box appears showing the area in which the text will be placed.

4. Type the following: **Mentor Graphics**

As you type, the characters are displayed in the text box. For this function, the RETURN key moves the text input point to the next line, so do not press the RETURN key.

5. Press the **OK** button on the prompt bar to complete the function. Do not worry if the text momentarily disappears when you move

the cursor out of the Edit window. When the prompt bar reappears, press the **Cancel** button to remove the prompt bar.

Figure 1-19 shows an example of the logo with text.

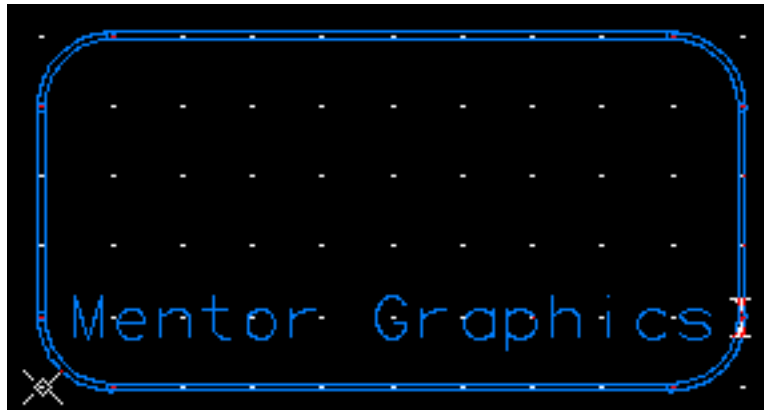


Figure 1-19. Logo with Text

The text appears as centerline text. To more accurately show the look of plotted text, change the view style.

6. Choose the **View > Change View Style...** menu item. Choose the Text Style: **Stroked**. Finally, press **OK**.

Adding the Shapes to the Logo

You now add some shapes to the logo that will be solid copper areas on the completed board. Refer to Figure 1-15 for a picture of the complete logo.

1. Choose the [**Top Menu**] **Shapes > Add Polygon:** menu item.

The prompt bar already has the location prompt highlighted.

2. Position the cursor at absolute coordinate 0.35, 0.45, and click the Select mouse button. Move the cursor to 0.6, 0.45, and click the Select mouse button again.

When you moved the cursor, you saw a ghost image of the first side of the polygon.

3. Move the cursor to 0.6, 0.2, and click the Select mouse button again.

Now you see three sides of the polygon, because that is the number of points you have defined so far.

4. Move the cursor to 0.35, .2, and click the Select mouse button a final time. **OK** the prompt bar to finish the square. Do not cancel the prompt bar, you will need it in step 6.

The square is completed. Next, you will create the triangle. You will be given only the coordinate location for the first point of the triangle.

By default, the cursor may only snap in orthogonal or diagonal directions. Before you add the triangle, change the cursor snap.

5. Choose the **Setup > Snap Direction...** menu item, and from the dialog box, choose: Select **Any Angle**. Finally, **OK** the dialog box.
6. Add the triangle to the left of the square using the same method you used to create the square. Start the triangle at absolute location 0.1, 0.2. Locate the top of the triangle at absolute coordinate 0.2, 0.45,

and locate the final corner at 0.3, 0.2. **OK** the prompt bar. **Cancel** the Add Polygon prompt bar when you finish.

Next you will add the circle.

7. Choose the [**Top Menu**] **Shapes > Add Circle > Center Points:** menu item.

The prompt bar already has the location prompt highlighted.

8. Position the center point of the circle by placing the cursor at absolute coordinate 0.8, 0.3, and clicking the Select mouse button. Next, move the cursor to see the white ghost of the circle. When the circle is the desired size, click the Select mouse button to complete the circle. **Cancel** the Add Circle Center Points prompt bar when you finish.

The circle is created with a double line so that the copper will form a circle, like an *O*, not a filled disk. If you want a filled disk instead of a circle, you can specify you want a line width of zero in the Add Circle Center Points options dialog box. Then when you create the circle, you will not see a double line as you do in this example; you would see a single line. The entire area of the circle would then be filled with copper.

The logo is now complete, as shown in Figure 1-20. To get a better idea of what the logo will look like, you will turn on the fill patterns in the next step.

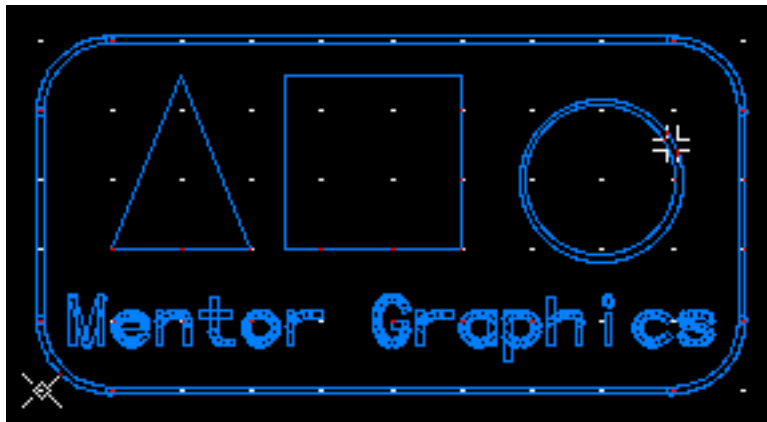


Figure 1-20. Completed Logo

9. Choose the **View > Change View Style...** menu item. In the dialog box, choose Polygon Style: **Filled**. Press the **OK** button in the dialog box.

Only the double line of the circle is filled, not the circle's center. If you had specified a zero line width in the Add Circle Center Points options dialog box, the circle would have been filled.

You save the logo geometry later in the lab.

Reading and Modifying Geometries

Instead of creating new geometries, it is sometimes easier to modify an existing geometry. This procedure demonstrates two techniques:

- How to read existing geometries from a library into the LIBRARIAN session.
- How to modify an existing graphic.

Next, you will read into LIBRARIAN a card ejector from the User library you copied as part of the training data, and you will modify the geometry by lengthening a portion of the ejector.

Creating Links to Geometry Libraries



If you are taking this training course as part of a workshop, skip this procedure section, and begin with procedure "Reading a Geometry from the User Library" on page 1-47. Create the links to geometry libraries only if you are completing this training as a Personal Learning Program. In the workshop version of this training material, the libraries are set up for you by your instructor.

1. Choose the **Geometries > Add Library Link...** menu item. In the dialog box that appears, enter the following, then press **OK**.

Library Name: **mgc.trng.hardware**

Pathname to Existing Library:

**your_path/training/board_new/mod3/sig_az/pcb_parts/
user_geom/hardware**

Add to: **User**

Directory type: **Permanent**

Enter the pathname carefully; pathnames are case sensitive. You have just created a link named *hardware* in your *\$HOME/pcb_parts/user_geom* directory that points to: *your_path/training/board_new/mod3/sig_az/pcb_parts/user_geom/hardware*. This menu item is equivalent to using the UNIX `ln -s` command in a shell.

Next you will create links to two other libraries. When you complete all the training, you will remove the links.

2. Choose the **Geometries > Add Library Link...** menu item, enter the following in the dialog box, and then **OK** the dialog box.

Library Name: **mgc.trng.padstacks**

Pathname to Existing Library:

**your_path/training/board_new/mod3/sig_az/pcb_parts/
user_geom/padstacks**

Add to: **User**

Directory type: **Permanent**

3. Add the final library link, using the same menu path you used previously, and enter the following values in the dialog box:

Library Name: **mgc.trng.components**

Pathname to Existing Library:

your_path/training/board_new/mod3/sig_az/pcb_parts/
user_geom/components

Add to: **User**

Directory type: **Permanent**

After you enter the library links, you do not need to type in the pathnames to access the libraries at any time later in the training.

Reading a Geometry from the User Library

1. Access the card ejector in the User library by choosing the **Geometries > List Geometry Libraries...** menu item.

A dialog box appears showing the predefined geometry libraries and indicates whether or not the libraries contain geometries.

2. Examine the dialog box and notice the names listed on the left side.

MENTOR
COMPANY
PROJECT
USER
NO DESIGN ACTIVE
OTHER GEOMETRY LIBRARIES
ALL GEOMETRY LIBRARIES

Each of the names represents a different level in the library hierarchy. If there is no data in the particular level, that level is dimmed. The purpose of this hierarchy is to help you organize and readily access the data on your network.

Next to the User level library, a note indicates that there are three libraries at the User level.

3. Move the cursor over **User** => **your_path/pcb_parts/user_geom** and click the Select mouse button.

The pathname following *User* might be slightly different on your workstation. The dialog box changes to show the data at the User level of the hierarchy, including the names of the three training libraries now available in the User level.

4. Move the cursor over **mgc.trng.components** and click the Select mouse button again.

The line highlights to show that the library is selected. The line also has the prefix *ACTIVE* => in front.

5. Select another library by placing the cursor on the library name and clicking the Select mouse button.

Another library is selected and made active. The first library is no longer active, but is still selected.

6. Place the cursor on the selected but inactive library and click the Select mouse button.

When you click the Select mouse button on an active library, it unselects. The library with the prefix *ACTIVE* => is used when you save a geometry to the active library.

7. If the **mgc.trng.components** library is not **ACTIVE**, make the **mgc.trng.components** library the active and only selected library. Then, at the bottom of the dialog box, press the **View** button.

The new dialog box lists the contents of the **mgc.trng.components** library. In the following steps you read the **card_eject** geometry from the **mgc.trng.hardware** library.

8. Return to the User Geometries Libraries dialog box by pressing the **User Libraries** button at the bottom of the Geometries List dialog box.

9. Select, activate, and view the **mgc.trng.hardware** library.

Now the contents of both the hardware library and the components library are listed at the same time.

10. From the **mgc.trng.hardware** library, select the geometry **card_eject**.

11. Press the **Read** button in the Geometries List dialog box.

The card ejector is read into the session and appears as a new Edit window. The edit window containing the logo is covered, but is not closed. You will learn how to save it later in this lab.

12. Activate the new Edit window by placing the cursor in that window and clicking the Stroke mouse button (you can optionally click the Menu mouse button, but you will briefly see the Popup menu).

Using Strokes

Strokes are a very fast and easy way to enter frequently used commands using the Stroke/Drag mouse button.

1. View the help information on strokes by placing the cursor in the edit window, holding down the Stroke mouse button (the center mouse button by default), and moving the cursor to draw a stroke in the shape of a question mark, as shown in Figure 1-21. (As you move the cursor, a line traces the movement of the mouse. This line is a *stroke*.)

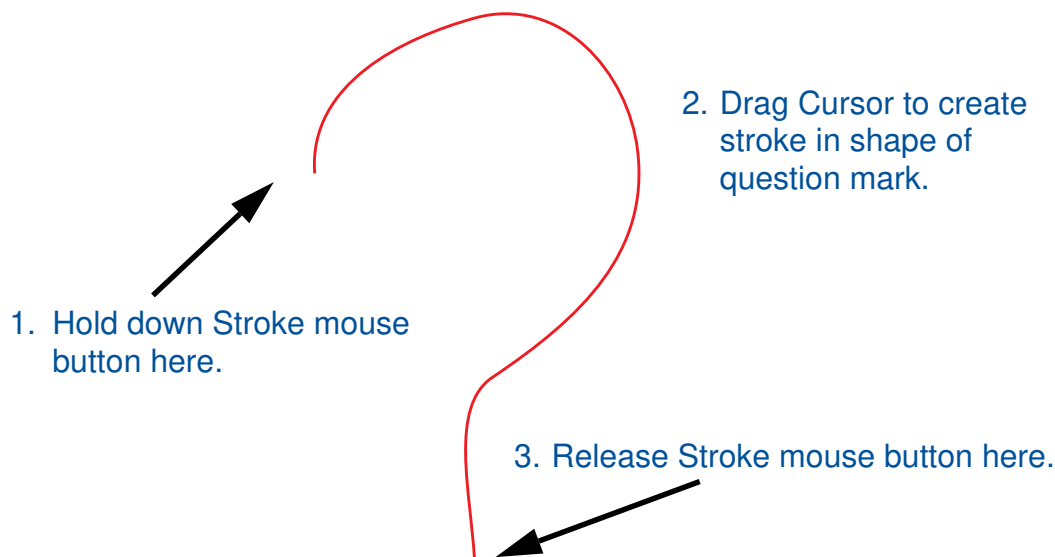


Figure 1-21. Stroke for Help on Strokes

A dialog box displays showing all the strokes provided by Mentor Graphics. Each picture of a stroke in the dialog box shows what function is performed for that stroke. For the viewing functions (view all, view area, and so on) that you need for this training, you will be directed to use strokes.

2. In the dialog box, notice the strokes you can use to change how the contents of the edit window is displayed. The upper-center portion of the dialog box shows the view strokes. Use the close window stroke, when you are done viewing the dialog box.

Next you experiment with the View All and View Area strokes.

3. View all of the card ejector by using the View All stroke anywhere in the edit window, as shown in Figure 1-22.

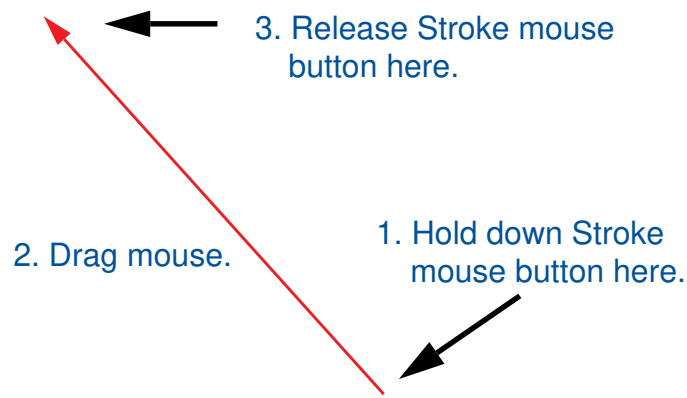


Figure 1-22. View All Stroke

The view in the edit window is changed so all of the geometry is visible. Next you will use the View Area stroke to view a close-up portion of the card ejector.

4. View only the area around the hole in the card ejector by using the View Area stroke over the hole as shown in Figure 1-23.

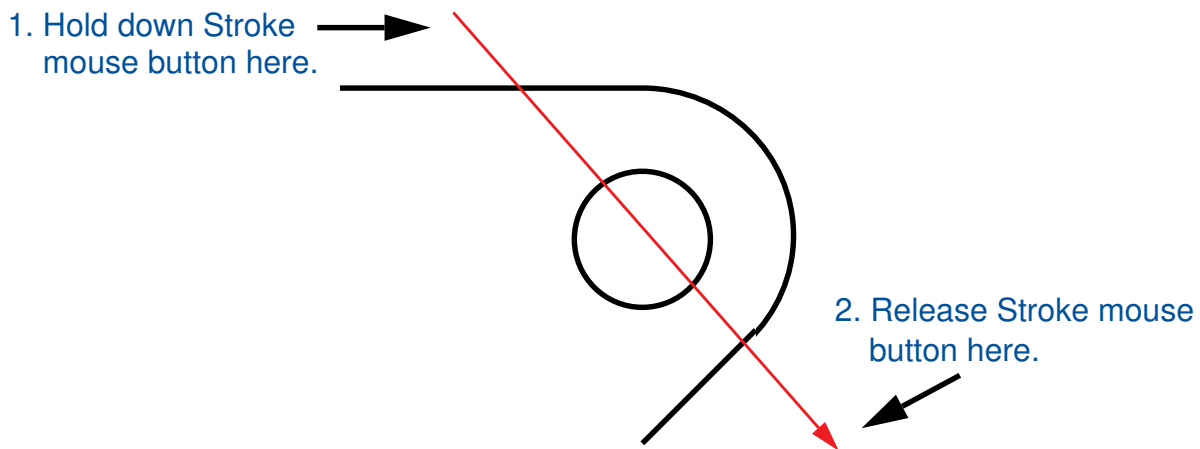


Figure 1-23. View Area Stroke

The view changes to show only the area between the beginning and end points of the stroke you drew.

5. View all of the card ejector again using the View All stroke.



If necessary, you can use the scroll bars along the edges of the Edit window to shift the image up and give a better view of the bottom tab of the ejector.

*If the scroll bars are not visible, you can display them by choosing the **Setup > Display Environment...** menu item, choosing **Show Scrolls** in the dialog box, and choosing **OK**. In the next steps, you lengthen the tab. Refer to Figure 1-24.*

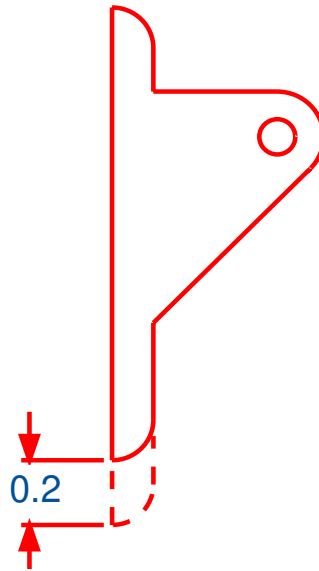


Figure 1-24. Card Ejector Geometry

6. Change the grid by choosing the **Setup > Grid...** menu item. In the dialog box, set the X Increment to **.025**. Set the Display Interval to **4**. Press the **OK** button in the dialog box.

The old grid, with its .05 X/Y increment, was too coarse to allow the modification needed. Changing the Display Interval from 2 to 4 while changing the grid increment results in an identical visible grid.

7. Choose the **Setup > Select Filter** menu item. In the dialog box, choose **Arcs** (so the button is darkened), and then **OK** the dialog box.

8. Place the cursor on the arc and click the Select mouse button.

The arc (and only the arc) turns white indicating that it is selected.

9. Choose the [**Top Menu**] **Shapes > Move** menu item.

10. Move the cursor and a ghost image of the arc follows. Position the ghost image 0.2 inches below the original arc (2 grid points to absolute coordinate location -0.5, -1.2) and click the Select mouse button.

The arc is re-positioned. You now need to stretch the lines down to meet the arc. The Select Area prompt bar remains on the screen.

11. Press the **Options** button in the prompt bar. In the Select Area (Options) dialog box that appears, choose **Vertices**, then **OK** the Options dialog box.

12. Place the cursor at the absolute coordinate location -0.6, -0.8, and hold down the Select mouse button. Drag the cursor (the select area is a dynamic rectangle) to the absolute coordinate location -0.3, -1.025 (until the rectangle encloses the endpoints of the lines), and release the Select mouse button.

When you release the Select mouse button, a white diamond appears at the end of each line indicating that the end vertices have been selected. The select count at the top of the Edit window reads 2.

The Move prompt bar automatically reappears.

13. Move the cursor in the Edit window and notice the white ghost *rubber bands* that follow the cursor. Position the lines so they connect with the arc that you moved, and click the Select mouse button. Cancel the Select Area prompt bar.

You have completed modifying the card ejector.

14. View the entire geometry, using the View All stroke.

Saving Geometries and Leaving LIBRARIAN

Now that these two geometries have been created, you need to save the geometries so that they can be used later. Since the logo is a new geometry and the card ejector is a variation of a master geometry and neither has been officially checked, it is not be a good idea to save them in the master libraries. A better place to save the geometries is in another library where you still have access to them while waiting for them to be checked and approved.

Saving Geometries in a Workshop

If you are completing this training in an instructor-led workshop, use this procedure section to save your geometries. If you are doing this training in a self-paced Personal Learning Program, skip this procedure and complete section "Saving Geometries in a Personal Learning Program" on page 1-55.

1. Choose the **File > Save > Save ASCII Geometries...** menu item. Fill in the dialog box as follows, then **OK** the dialog box:

Geometries to Save: **All Geometries**
Separate File For Each
Library to Store the Geometry: **Other**
Directory Pathname: **your_path/pcb_parts/user_geom/trng**
Replace Existing File(s)

When you need to retrieve the two files (logo and card_eject) from your new trng library later, you create a link to this new trng library.

You chose to replace existing files even though you are creating a new directory (trng). You did this just to make certain the new directory is created.

A Report-Message window appears confirming that the files were written and that they were written to the correct location.

2. Close the report-message window.

Saving Geometries in a Personal Learning Program

Complete this procedure section only if you are doing this training as part of a self-paced Personal Learning Program.

1. Choose the **File > Save > Save ASCII Geometries...** menu item. Fill in the dialog box as follows, then **OK** the dialog box:

Geometries to Save: **All Geometries**

Separate File For Each

Library to Store the Geometry: **Other**

Directory Pathname:

your_path/**training/board_new/mod3/sig_az/pcb_parts/
user_geom/trng**

Replace Existing File(s)

There is a *pcb_parts* directory that is provided for you in your training design data directory. You wrote the trng file in that directory instead of into your *\$HOME/pcb_parts* directory so that the training data is not mixed up with your permanent *pcb_parts* directory. The system records the name, creates the directory, and saves the geometries at the directory pathname you specified.

When you need to retrieve the two files (logo and card_eject) from your new trng library later, you create a link to this new trng library.

You chose to replace existing files even though you are creating a new directory (trng). You did this just to make certain the new directory is created.

A Report-Message window confirms that the files were written and to the correct location.

2. Close the report-message window.

Closing LIBRARIAN

Close the LIBRARIAN session by choosing **Close** from the Window Menu (upper-left corner icon). When the *Save change to design?* dialog box appears, choose **No**.

You choose no because:

- You have already saved the geometries to a **trng** directory where they can be accessed for any new design.
- You did not enter LIBRARIAN with a design so there is no active design in which to save them.

Congratulations! You have completed the "Introduction to LIBRARIAN" lab exercise. Continue with Lesson 2: "Basic Geometry Creation Techniques".

Lesson 2

Basic Geometry Creation Techniques

In this lesson, you study basic techniques for creating graphics for components. In Figure 2-1, you can see where geometry creation fits with the overall process of circuit board design.

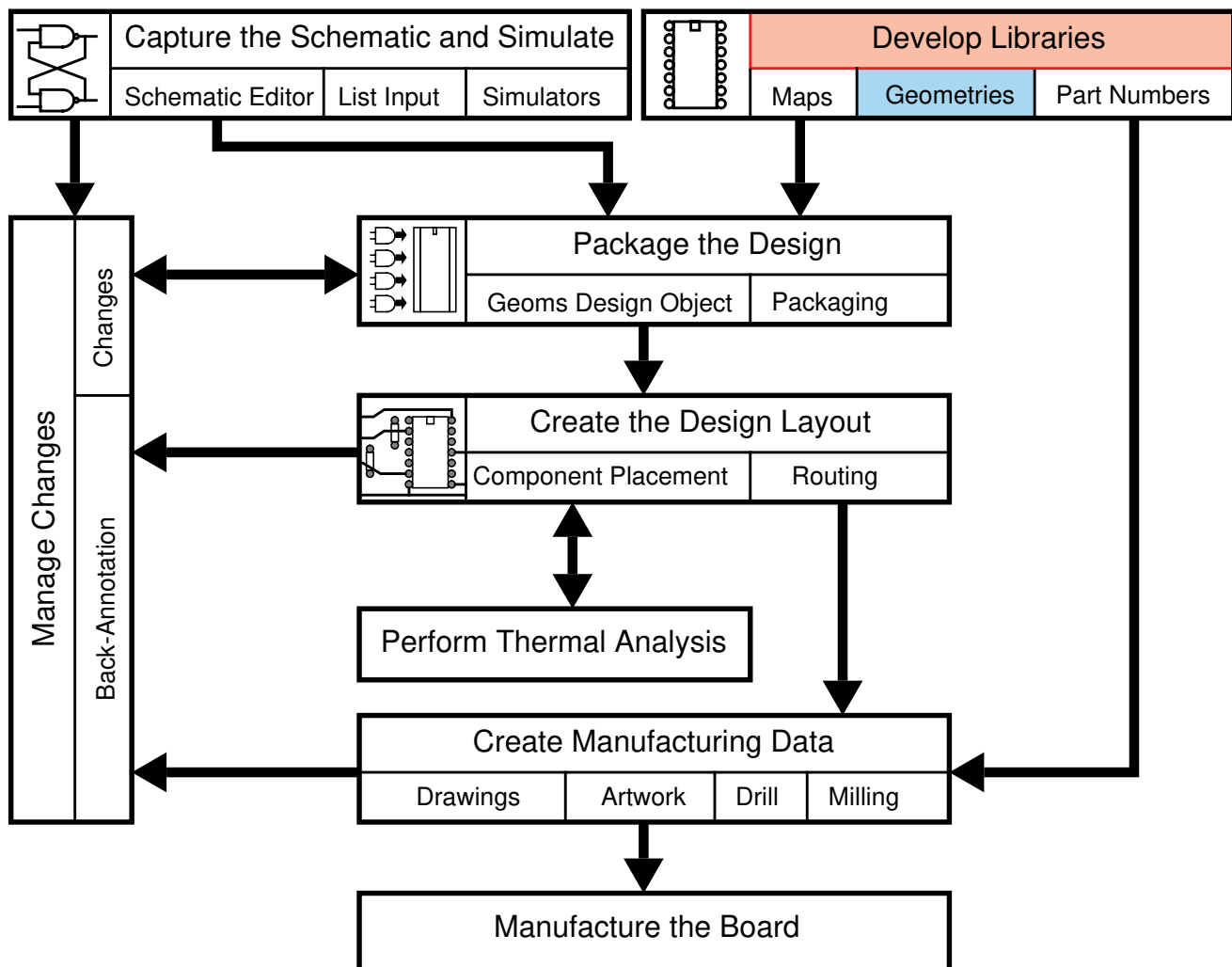


Figure 2-1. PCB Design Process

Objectives

In the previous lesson, you learned a few very basic methods of creating geometries. In this lesson, you build on your knowledge so that you learn to construct most types of geometries. After this instruction, you can experiment on your own to create any geometry you might need. At the completion of this lesson and its associated lab exercise, you can:

- Set up an appropriate grid.
- Set up edit and viewing layers for geometry creation.
- Create, edit, move, and copy graphic elements in either the absolute coordinate system or the delta coordinate system.
- Determine when to use the absolute coordinate system or the delta coordinate system.
- Create lines, polygons, circles, and arcs.
- Use the Select Filter to control what types of objects can be selected for editing.
- Edit existing geometries by adding fillets or trimming.
- Add or position graphic elements relative to the location of an existing graphic element.

Setting up the Editing Environment

Before you create geometries, you set up the editing environment using menu items in the **Setup** pulldown menu. There are many conditions you can set to determine your editing environment, such as line style and width, text style, selection filters, editing and viewing layers, and the grid. For a complete discussion, refer to section "Setting the Design Environment" in the *LIBRARIAN User's Manual*. In this lesson, only the basic and most important features are discussed to provide you more time to experiment with the features during the lab session.

Edit Layers

In LIBRARIAN and FabLink, all graphics are contained on layers. There is a specific layer for each type of graphic object. For example, there are edit layers for component body outlines, dimensions, signal traces and power planes, the board outline, and so on. You do not use all the layers in LIBRARIAN, because information is added to some layers in other applications, such as LAYOUT.



In LAYOUT, some graphic elements display as objects only.

Before you create graphics, you specify on which layer the data is to be placed. You specify the edit layer by choosing the **Setup> Edit Layer** menu item and select the edit layer from the Set Edit Layer dialog box, as shown in Figure 2-2.

The edit layers available are the same as the names of the view layers. The edit layer must also be one of the layers you specify to view. The currently set edit layer name displays in the upper-right corner of the edit window. The default edit layer is SIGNAL_1.

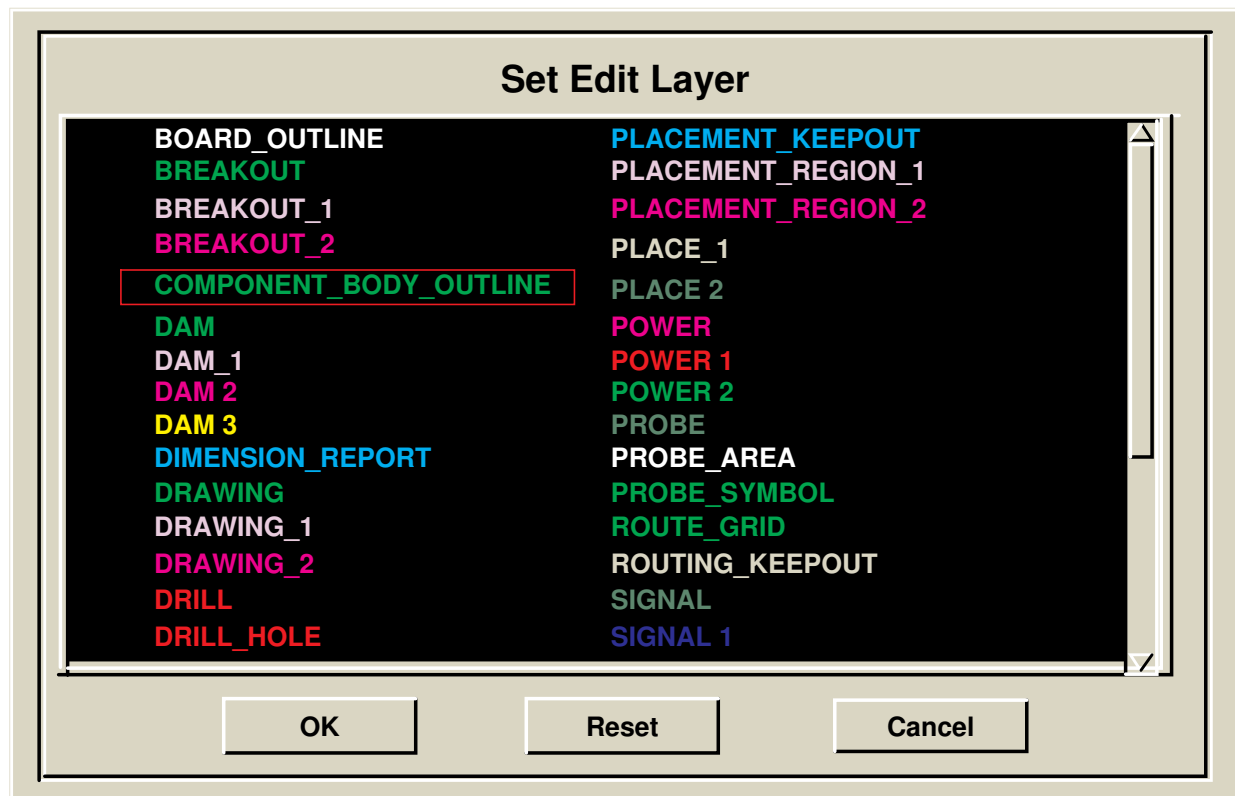


Figure 2-2. Set Edit Layer Dialog Box

View Layers

Before you start editing, make sure the edit layer you have specified is also one of the viewed layers. Using the **View > Layers** pulldown menu item, you can specify which of the layers is visible. By default, all layers are visible. However, if there is a lot of data on several layers, and all layers are visible, seeing any detail of a single layer becomes difficult. Therefore you can limit which layers are visible to make viewing easier. The View Layers dialog box looks much like the Set Edit Layer dialog box except that each layer name in the list has a *V* and/or an *S* next to it. The *V* indicates the layer is visible, and the *S* indicates that graphic data on that layer can be selected. Using options in the dialog box, you can change the visibility and selectability of data on each of the layers.

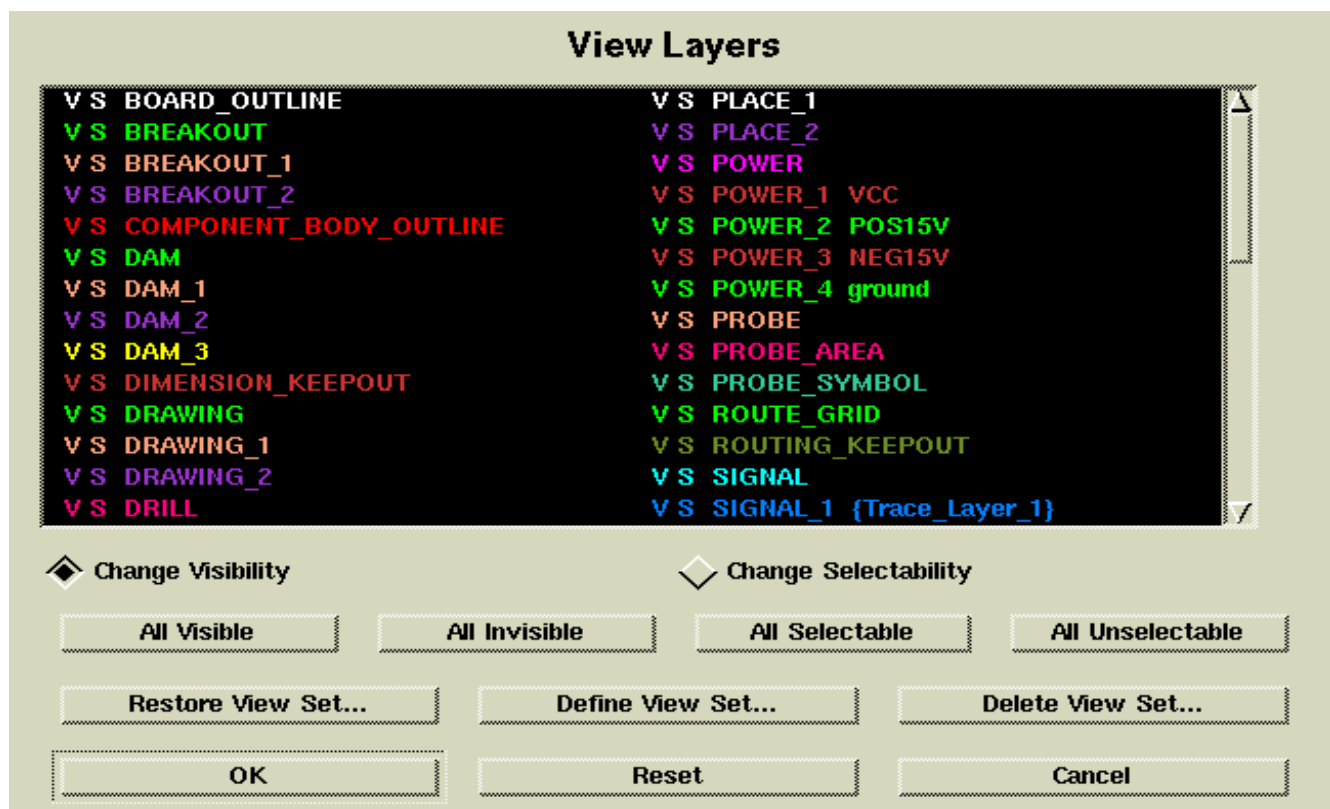


Figure 2-3. View Layers Dialog Box

The Display Grid

The display grid is made of intersecting, regularly spaced horizontal and vertical lines in the Edit window. You use the grid to position graphic shapes at specific coordinate locations. The settings for the display grid determine the spacing between the vertical lines (x-axis) and for the horizontal (y-axis) lines. You can specify different spacings for the x and y axes, if needed. Grid points mark the intersections of the grid lines. The grid lines are not visible. Only the grid points are visible. If you want, you can specify an interval value, which determines how many of the possible grid points are visible. For example, an interval of 2 specifies that every other grid point is visible. The banner of the Edit window reports the current grid setting.

To change the grid display, choose the **Setup > Grid** pulldown menu, and fill in the Set Grid dialog box, which is shown in Figure 2-4.

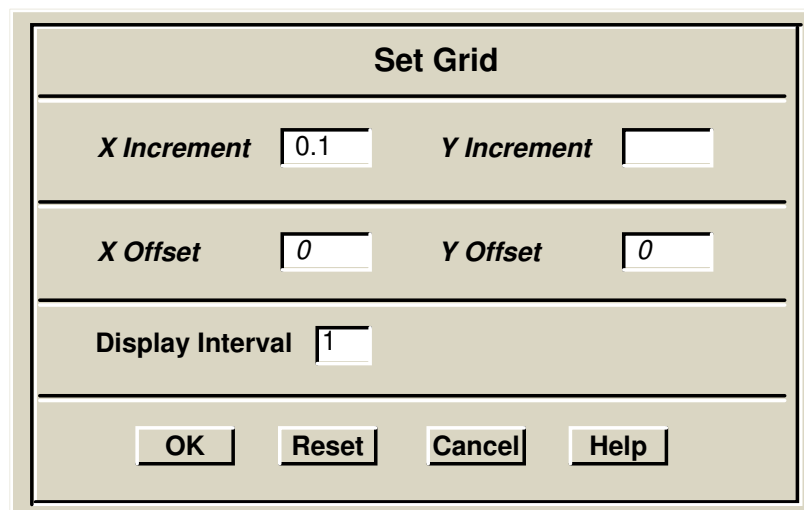
The image shows a 'Set Grid' dialog box with a tan background and a black border. It contains three rows of input fields. The first row has 'X Increment' with a value of '0.1' and 'Y Increment' with an empty field. The second row has 'X Offset' with a value of '0' and 'Y Offset' with a value of '0'. The third row has 'Display Interval' with a value of '1'. At the bottom, there are four buttons: 'OK', 'Reset', 'Cancel', and 'Help'.

Figure 2-4. Set Grid Dialog Box

Grid Visibility and Grid Snapping

Grid visibility refers to whether or not the grid points are visible. If the grid is currently not visible, and you want it to be visible, choose the **View > Grid On** menu item. If the grid is visible, and you do not want it to be visible, choose the **View > Grid Off** menu item.

Grid snapping makes the cursor and graphic data snap to the grid points. If snapping is on, the cursor and graphics snap to all grid points, not just the visible points. Grid snapping is useful for creating graphics of a specific size and with correct and consistent pin spacing. If grid snapping is off, graphics can be placed off grid, but it becomes difficult to create exactly sized graphic data. For example, having grid snapping off is useful for placing text off the grid. You turn the grid snapping on or off using the **Setup > Grid Snap On** and **Setup > Grid Snap Off** menu items.

Selecting Geometry Data

In all Mentor Graphics applications, you must select an object before you can edit it. After you edit something, you must unselect it to prevent further unwanted edits.

The easiest way to select something is to place the cursor on it, and click the Select mouse button. If you want to select several objects at the same time, you first imagine a rectangle that encloses all the objects you want to select, place the cursor at a corner of your imaginary rectangle, hold down the Select mouse button and move the mouse. As you move the mouse, a rectangle is formed. You continue moving the mouse until all the objects you want to select are within the rectangle you form by moving the mouse. Finally, you release the Select mouse button. The rectangle formed by moving the mouse is removed, and all the objects that were within the rectangle are selected. The number of items selected is displayed in the upper-left corner of the session window, as shown in Figure 2-5.

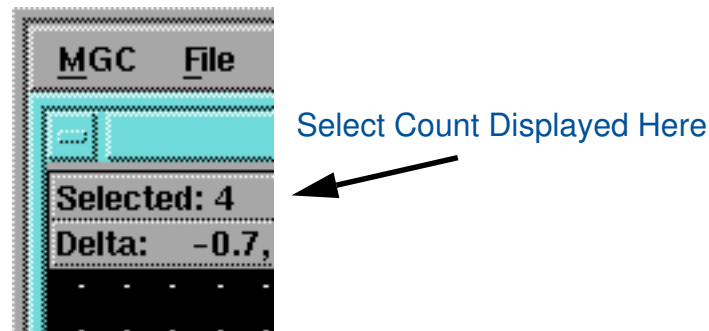


Figure 2-5. Where to Locate Number of Selected Items

It is important that you are always aware of how many objects are selected so that you do not unexpectedly edit or delete something. You can use the *Selected:* count for selecting, editing, unselecting, and verifying the status of all objects.

There are other ways, besides using the mouse, to select graphics and text. You can use the Select Area function key together with the mouse, instead of using the Select mouse button. You can also use any of the **[Shapes] Select** submenu items.

Unselecting Geometries

To unselect items you do not want to edit, you can either use the Unselect All function key, which is probably the most convenient method of unselecting everything, or you can use the **[Shapes] Unselect > Unselect All** menu item. If you want to unselect only one or a few items, leaving others selected, you can use the Unselect Area function key together with the mouse using the same method as for selecting items in an area. You can also use the **[Shapes] Unselect > Unselect Area** menu item to define the Select mouse button to unselect objects.

Select Filter and Unselect Filter

Sometimes when you are editing complex geometries, you need the ability to filter what can and cannot be selected and unselected. Using the **Setup > Select Filter** pulldown menu item, and its associated dialog box as shown in Figure 2-6, you can determine what kinds of objects can and cannot be selected. The **Setup > Unselect Filter** and its dialog box determines what can or cannot be unselected.

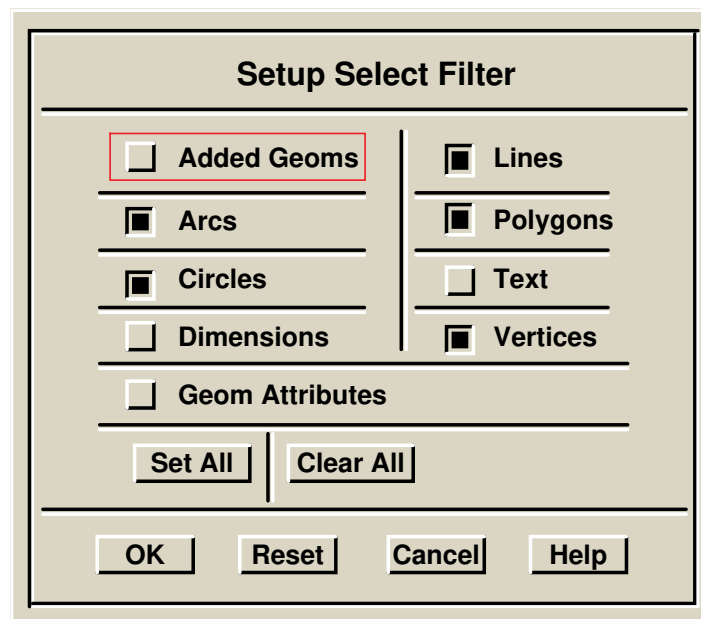


Figure 2-6. The Setup Select Filter Dialog Box

These are useful, for example, when there is a lot of data in a small space, and you want to select only the text for editing. In other cases, you might find it easier to select all objects, and then only unselect some of the objects so that the remainder can be edited. Using the select and/or unselect filters you can specify that any combination or set of Added Geoms, Arcs, Circles, Dimensions, Geom Attributes, Lines, Polygons, Text, and Vertices can be selected or unselected. By default, all of these objects are selectable and unselectable.

Coordinate Systems

To help you accurately position graphics in the Edit window, LIBRARIAN provides three coordinate systems; The Absolute coordinate system, the Delta coordinate system, and the Polar coordinate system. In this training we cover only the Absolute and Delta systems. The Absolute and Delta coordinate systems each have their own origin. The current cursor location is displayed in both coordinate systems simultaneously in the left end of the Edit window banner, as shown in Figure 2-7.

Delta: -1.59045, 0.5458 Abs: -1.4907, 0.7424

Figure 2-7. Current Cursor Coordinate Locations

Absolute Coordinate System

The absolute coordinate system is based on the origin at the center of the Edit window. All coordinate locations entered in this coordinate system are relative to this origin. The origin appears at the center of the Edit window, and looks like an X with a diamond in the center, as shown in Figure 2-8.

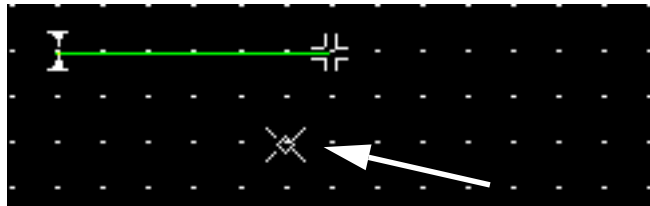


Figure 2-8. Origin of the Absolute Coordinate System

The first graphic element you create for a component is located relative to the Absolute coordinate system. The location of the origin of the Absolute coordinate system, relative to the graphics, is important when you place the component in LAYOUT. Locate the origin either in the center of pin 1 of the component, or place the origin in the center of the component. When you move the cursor to position a component in LAYOUT, the component is oriented relative to its origin in LIBRARIAN.

When you are creating graphics, and you are prompted for a location, to enter a coordinate in the Absolute coordinate system, choose the **[Shapes] > Snap > Absolute** popup menu item and enter the coordinate in the prompt bar. When you OK the prompt bar, the coordinate point is entered. You can enter an Absolute coordinate any time you are prompted for a location.

Delta Coordinate System and the Basepoint

The Delta coordinate system can use for its origin either the basepoint, or the lastpoint. The basepoint is shown in Figure 2-9. The lastpoint is shown in Figure 2-10. When you are creating graphics, you can place the basepoint anywhere, and then add the graphics relative to the basepoint. For example, you can place the basepoint on vertices, centers, or midpoints of existing graphic data. Using the basepoint in this way makes it easy to enter coordinates, because you don't have to calculate coordinates relative to the Absolute origin. Instead, you can place the graphic data relative to another geometry.

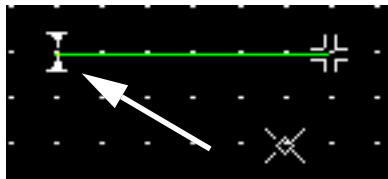


Figure 2-9. Origin of the Delta Coordinate System

In addition to the basepoint, you can use the Delta coordinate system to add graphics relative to the lastpoint. The lastpoint is an icon automatically located on the last vertex of the most recently added geometry. Therefore, when you create graphics and you enter coordinates in the Delta coordinate system, you can enter a location relative to the last vertex's location. For example, to specify the location for the next vertex of a rectangle using the Delta coordinate system and the lastpoint, you need only enter the length of the side (such as .5,0 for a .5 inch long horizontal side). In the Absolute coordinate system, you would have to specify where that point is relative to the Absolute origin, and the required coordinate might be difficult to determine.

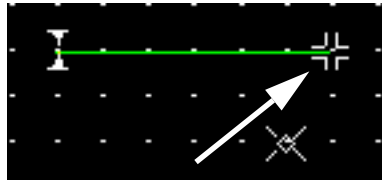
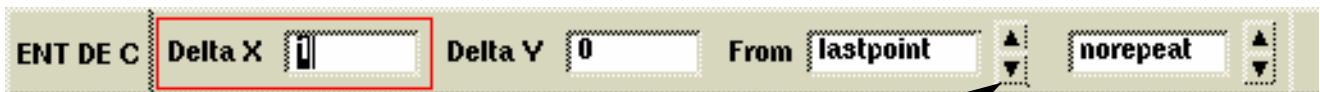


Figure 2-10. Lastpoint Icon

When you are adding graphics, and you are prompted for a location, you can specify a coordinate in the Delta coordinate system by choosing the **[Shapes] > Snap > Delta** popup menu item. The Enter Delta Coordinate prompt bar displays, as shown in Figure 2-11. To specify the coordinate, you enter the X and Y values, and then choose which origin you want the X and Y values to work from. You can choose either the lastpoint, which is the last vertex specified when creating graphic data, or you can specify the basepoint, which you can place anywhere before you enter the Delta coordinate.



You choose the origin of the Delta coordinate system by clicking on the up/down scroll arrows to show either *lastpoint* or *basepoint*.

Figure 2-11. Enter Delta Coordinate Prompt Bar

Snapping

Snapping is a term used to describe placing a basepoint or defining a location that is on existing graphic data without having to know the coordinates of that point. For example, you can snap to endpoints, midpoints, and intersections of lines, or the center of circles. The possible locations for snapping are listed in the **[Shapes] Snap** submenu, shown in Figure 2-12. This makes creating new graphic data or moving or copying existing graphics very simple, because you do not have to calculate any coordinate points.

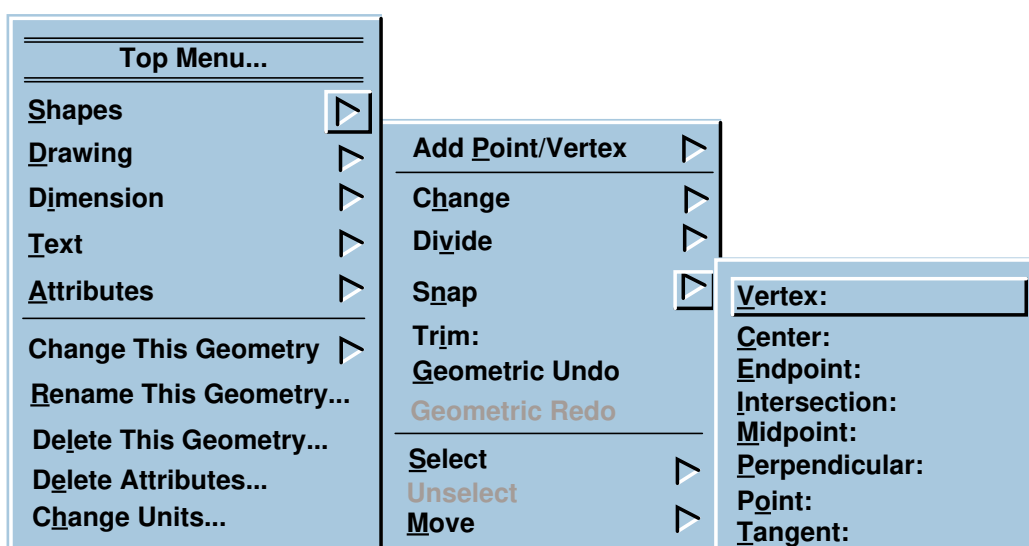


Figure 2-12. Snap Submenu

If you want to move or copy graphic data relative to the location of other graphics, the easiest way to do it is to snap the *From* point to a convenient location on the graphics you want to move or copy, and then snap to a convenient point on other graphic data to specify the *To* point.

For example, if you wanted to move a line so that its midpoint is relocated on the endpoint of another line, you could use the following procedure:

1. Select the line you want to move.
2. Choose the **[Shapes] Move > Move From To** popup menu item.

3. When you are prompted for a From location, choose the **[Shapes] Snap > Midpoint** menu item, place the cursor on the selected line, and click the Select mouse button.

The *From* point is now the midpoint of the selected line, and the prompt bar changes to prompt you for the *To* point.

4. At the To prompt, choose the **[Shapes] Snap > Endpoint** menu item, place the cursor on the end of the line to which you want to move the selected line, and click the Select mouse button.

The selected line is moved so that its midpoint is on the endpoint of the other line.

You can use this same general procedure to move or copy graphics to any location on other graphics, using the various snapping locations available.

Another use for snapping, which you might use more often, is to move the basepoint by snapping it to existing graphics so that you can create new graphic data relative to the new basepoint location.

For example, you might want to create a circle (for a drill hole) near the intersection of two lines (the corner of a board). To do this you could use the following procedure:

1. Choose the **[Shapes] Move > Basepoint** popup menu item.
2. Choose the **[Shapes] Snap > Intersection** popup menu item, then place the cursor on one of the lines forming the intersection and click the Select mouse button. Next, place the cursor on the other line, and again click the Select mouse button.

The basepoint is placed on the intersection

3. Choose the **[Shapes] Add Circle > Center Value** menu item. In the prompt bar, enter the radius of the circle you want, then press the Tab key to highlight the Location prompt.
4. Choose the **[Shapes] Snap > Delta** popup menu item, and enter the distance, in X and Y coordinates, from the basepoint you placed at the intersection of the lines. Also choose **basepoint** in the From prompt, and **OK** the prompt bar.

The circle is added at the location you specified from the intersection of the lines.

Creating Geometries

You were introduced to some of the basic features of creating graphic data during the previous lab exercise. You experienced creating lines, arcs, circles, and polygons, and you copied and flipped graphics. In this section of the lesson, you review those features, and you are briefly introduced to other kinds of graphics that you can create.

Adding Lines

In addition to the simple two-vertex lines you created in the previous lab exercise, you can create other kinds of lines. The kinds of lines you can create are listed in the Add Line submenu, shown in Figure 2-13.

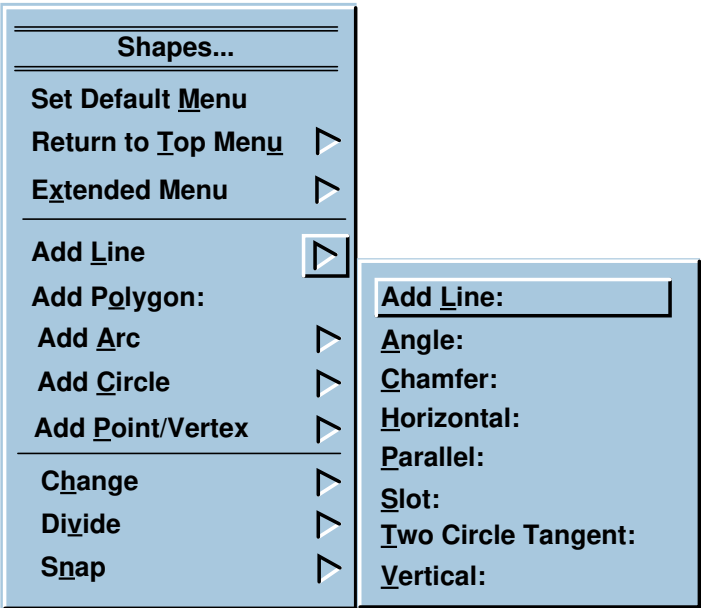


Figure 2-13. The Add Line Submenu

The [Shapes] Add Line > Add Line menu item presents you with a prompt bar, as shown in Figure 2-14, that prompts you for points through which the line will pass. When you see an icon-prompt that appears like the *points* prompt in the Add Line prompt bar, where the location points appear stacked, it means that you can specify two or more location points. Because you see this prompt in the Add Line prompt bar, you can specify a line that passes through many vertices. You can specify the points by clicking on points with the Select mouse

button, by entering coordinates in any coordinate system, or by snapping to existing graphic data.

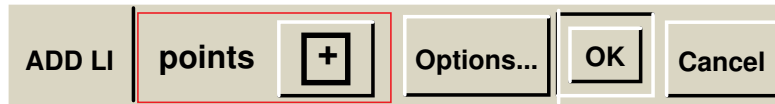


Figure 2-14. Add Line Prompt Bar

The **[Shapes] Add Line > Angle** menu item creates a line through any point you specify, at any angle. The resulting line extends to the edges of the currently viewed area. If you want a long line, first view a large area before you create the line.

The **[Shapes] Add Line > Horizontal** and **[Shapes] Add Line > Vertical** menu items create horizontal or vertical lines, respectively, through a single point that you specify. The lines extend to the edges of the currently viewed area.

The **[Shapes] Add Line > Parallel** menu item creates a line parallel to an object that you select, and through a point you specify. The resulting line extends to the edges of the currently viewed area.

To use the Angle, Horizontal, Vertical, and Parallel line features to create useful shapes, you can place the lines on points so the lines enclose an area in the shape of the graphics you want to create, leaving the ends of the lines extending beyond the shape to any arbitrary lengths. An example of this is shown in Figure 2-15. Then, using the **[Shapes] Trim** menu item, you can easily trim the ends of each line back to where they intersect another line, leaving only the outline of the graphics, as shown in Figure 2-17. Instead of trimming the lines back to the intersections, you could make fillets, which can automatically trim the ends off the lines.

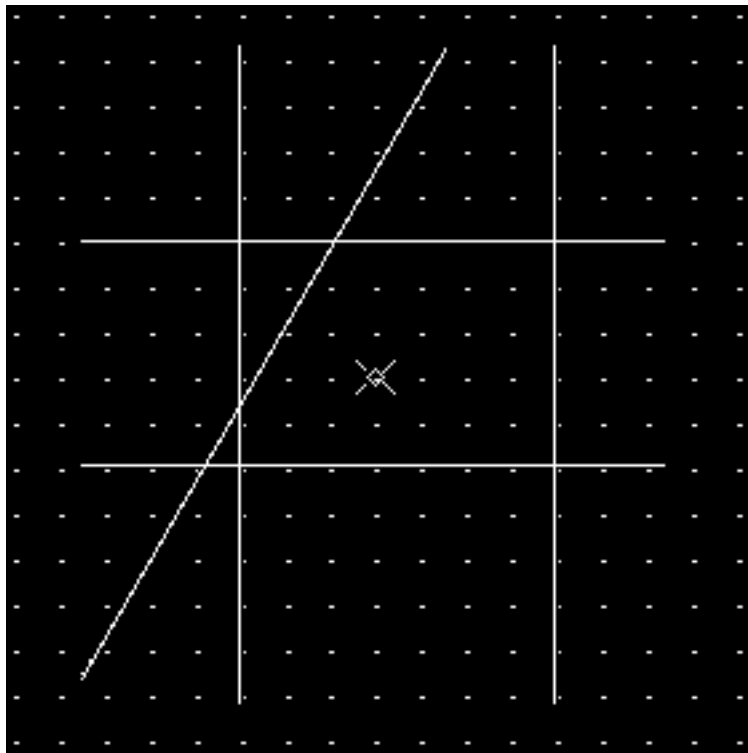


Figure 2-15. The Outline of a Polygon Defined by Placing a Series of Lines

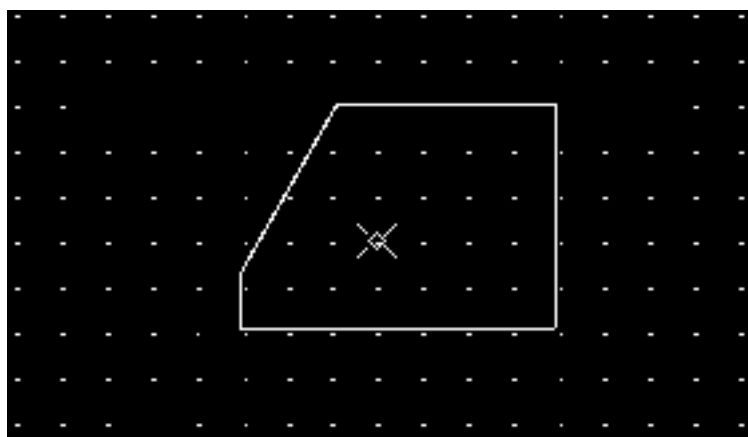


Figure 2-16. The Polygon After Trimming the Ends of the Lines

Adding Polygons

Another way to create polygons, rather than placing lines and trimming their ends, is to use the **[Shapes] Add Polygon** menu item. As when adding a line, you are prompted for a series of points on which the vertices of the polygon are placed. You can specify the points by clicking the Select mouse button in the Edit window, by entering coordinates in any coordinate system, or by snapping to existing graphics. You created polygons using this menu item in the previous lab exercise.

Polygons are always filled with copper on the board. If you create a polygon with a fill pattern, the fill pattern is for display only. Polygons with fill patterns, like all polygons, are filled with copper on the board.

Adding Circles

Just as there are many ways to create lines, so are there several ways to add circles. The menu items for creating circles are shown in Figure 2-17.

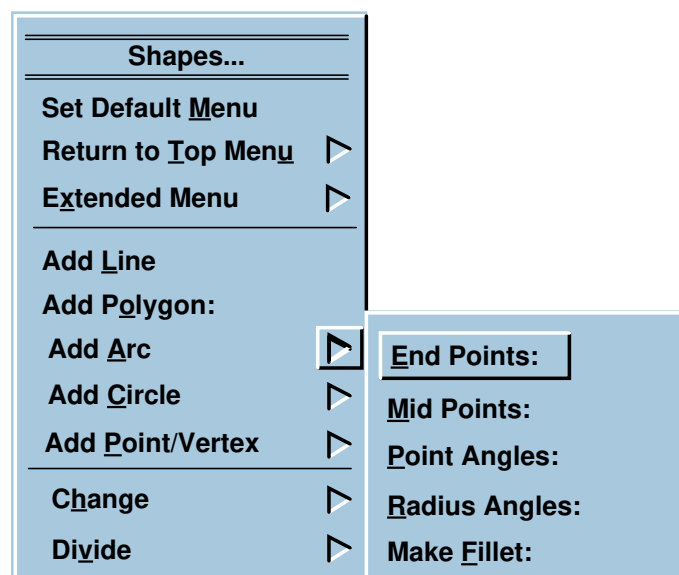
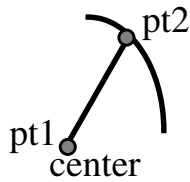
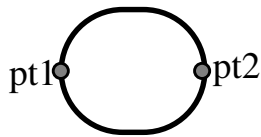


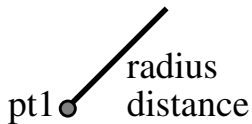
Figure 2-17. The Add Circle Submenu



The **[Shapes] Add Circle > Center Points** menu item allows you to create a circle by specifying the center point and a point on the circle. This is a good choice for creating circles if you know the location of the center point and the value of the radius, or if you want to create a circle of known radius that is tangent to another object. As with lines or any other graphics, you can specify the points by clicking on any location with the Select mouse button, by entering coordinates in any coordinate system, or by snapping to existing graphics.



The **[Shapes] Add Circle > Diameter Points** menu item creates a circle between two points you specify. This is a good choice for creating a circle if you know the diameter required, and the center point is not important. You can also use this to conveniently create circles that are tangent to other graphics.



The **[Shapes] Add Circle > Center Value** menu item prompts you for the value of the radius and for the center point location.



The **[Shapes] Add Circle > Circle to Arc** menu item prompts you to select an arc. When you select an arc, a circle is superimposed over the arc using the radius of the arc. By setting the select filter, you can select the arc and the circle individually and move, copy, or delete them.

Adding Arcs

As with other graphics, there are many ways to create arcs, as shown in the **[Shapes] Add Arc** submenu in Figure 2-18.

Of these menu items, one that you might use frequently is the **[Shapes] Add Arc > Make Fillet** menu item. This menu item provides a prompt bar that prompts you for a radius for the fillet, and a *From* and a *To* point. If you had a corner of a rectangle, for example, you start making a fillet at the corner by choosing the menu item, and entering the radius value. Then you place the cursor on either line forming the corner of the rectangle, and clicking the Select mouse button. This defines the *From* point. Finally, you place the cursor on the other line forming the corner, and click the Select mouse button again. A fillet of the specified radius is added, and the ends of the lines that formed the corner are automatically trimmed back to end at a tangent to the fillet. If you want, you can turn off the automatic line clipping.

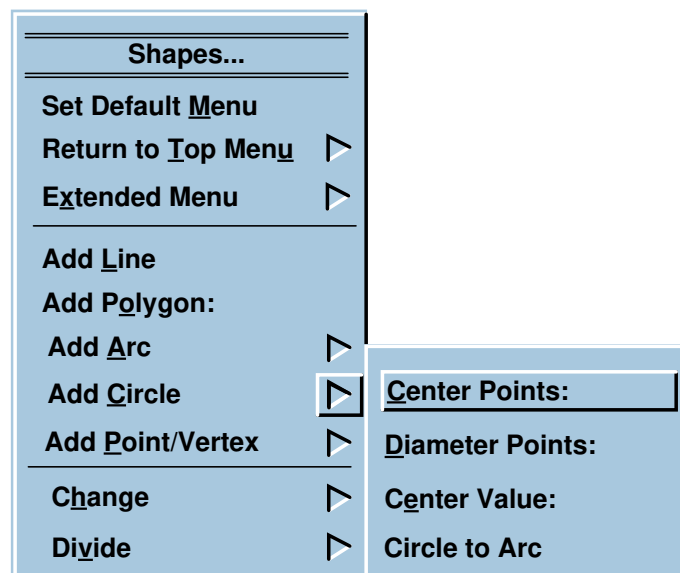


Figure 2-18. The Add Arc Submenu

Deleting Geometries

To delete graphics, you first select it, using the selection techniques described earlier in section “Selecting Geometry Data” on page 2-7. When the graphics are selected, choose the **[Shapes] > Delete** popup menu item. The **OK to Delete** dialog box displays, as shown in Figure 2-19. If the object you selected is something other than a set of vertices (text, for example), choose Other, and then OK the dialog box. The selected item is deleted.



Figure 2-19. The OK to Delete Dialog Box

If you see that you have accidentally deleted something, and you want to restore it, choose the **[Shapes] > Geometric Undo** menu item.

Moving and Copying Geometries

There are several methods for moving and copying geometries. This section describes the most basic method. The process for all types of moves and copies is to:

1. Select the geometry you want to move or copy, using the selection techniques described earlier in section “Selecting Geometry Data” on page 2-7.
2. Move or copy the selected geometry using either the menu items in the **[Shapes] > Move** or **[Shapes] > Copy** popup menus, or by using the Move or Copy function keys. You can also use the move or copy strokes, functions, or Shapes Palette menu icons.



In this training, you will be instructed to use the popup or pulldown menus for most functions. Sometimes, as when selecting or unselecting geometries, you will use the function keys. For viewing either all of an object or an area of an object you will learn to use strokes. You can use any method (menus, strokes, function keys, functions, or palette menu icons) to complete any task, but it is easier to learn new products when you concentrate on learning the process, rather than on learning all possible methods to complete a task.

3. After you complete the move or copy, unselect all geometries. The most convenient method of unselecting all geometries is to use the Unselect All function key.

If you make a mistake with the move or copy, you can undo it by using the **[Shapes] > Geometric Undo** menu item.

Basepoint Location

When you select a geometry using any method, LIBRARIAN places a basepoint somewhere on the geometry.

- Selecting a line places the basepoint at the left-most vertex. If the line is vertical, the basepoint is on the lowest vertex.
- Selecting a polygon places the basepoint at the lower-left vertex.
- Selecting a circle places the basepoint at the circle's center.
- Selecting an arc places the basepoint at the lowest, left point of the arc.

The location of the basepoint is important. When you move or copy a geometry, you specify a location where the basepoint of the moved or copied geometry will be placed. The orientation (distance and direction) of the moved or copied geometry to its basepoint is the same as the orientation of the original geometry to its basepoint. .

The following example and illustrations show how the basepoint relates to the geometry during moves and copies. To begin the move (or copy) process, you first select the geometry. You can select the geometry by clicking on it with the Select mouse button. If you forget to select the geometry, and you choose the **[Shapes] > Move** (or **[Shapes] > Copy**) menu item, you are prompted to select something. When you select the geometry, a basepoint is placed on the geometry, as shown in Figure 2-20.

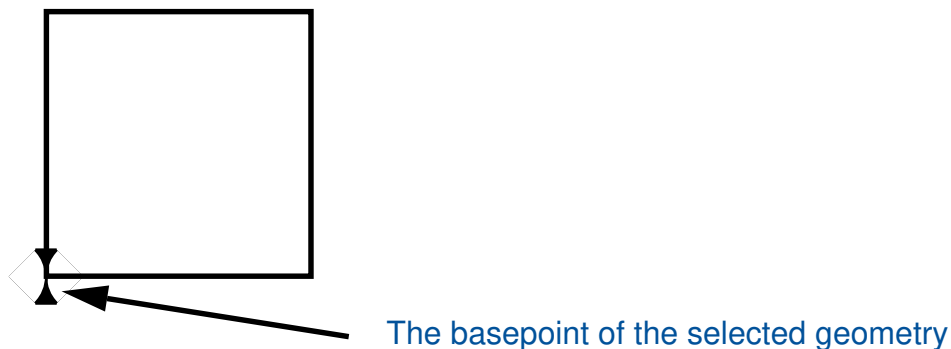


Figure 2-20. Selected a Geometry, and its Basepoint

When you have selected a geometry, and you choose the **[Shapes] > Move** menu item, you see the Move prompt bar. In the Move prompt bar, you see the Location prompt, as shown in Figure 2-21.

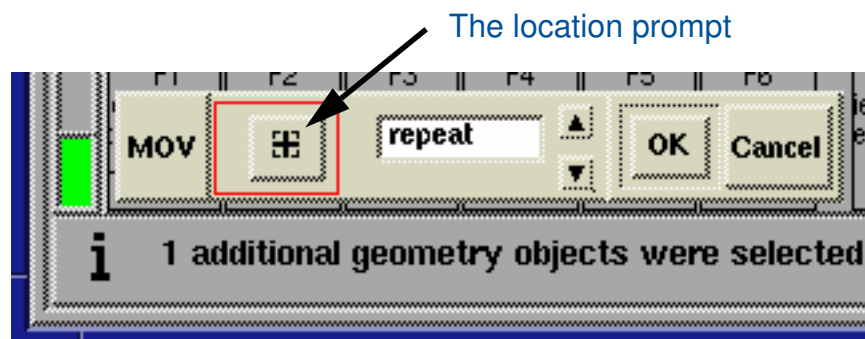


Figure 2-21. Move Prompt Bar and the Location Prompt

You must satisfy the location prompt by providing a location where you want the selected geometry to be moved. You can specify the location either by moving the mouse in the edit window and clicking the select mouse button (in which case you see a ghost-image of the selected geometry to help you position the geometry), or you can snap to any other existing geometry, or you can specify a coordinate location in any coordinate system. In this example, a location is specified in the Delta coordinate system.

With the Move prompt bar still visible, you choose the **[Shapes] > Snap > Delta** menu item. The Enter Delta Coordinate prompt bar is displayed over the top of the Move prompt bar. You enter the coordinate location, and specify *basepoint* in the From prompt, as shown in Figure 2-22.

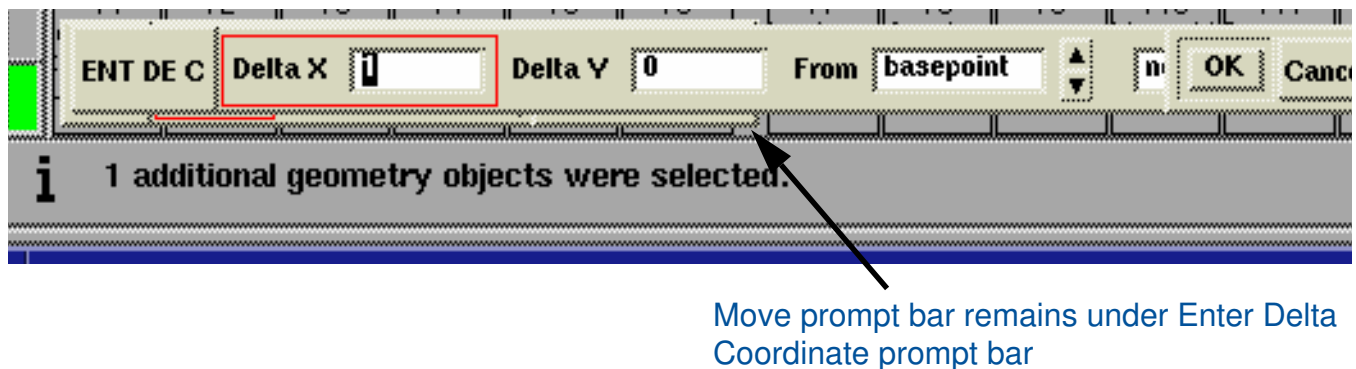


Figure 2-22. Enter Delta Coordinate Prompt Bar

Finally, you click on **OK** in the Enter Delta Coordinate prompt bar. Both prompt bars are removed because you have satisfied all the requirements of both prompt bars. The value of the coordinate in the Enter Delta Coordinate prompt bar satisfied the requirement of a location of the Move prompt bar.

The Delta Coordinate value you enter specifies where to move the basepoint of the selected geometry. The geometry follows along, as shown in Figure 2-23.

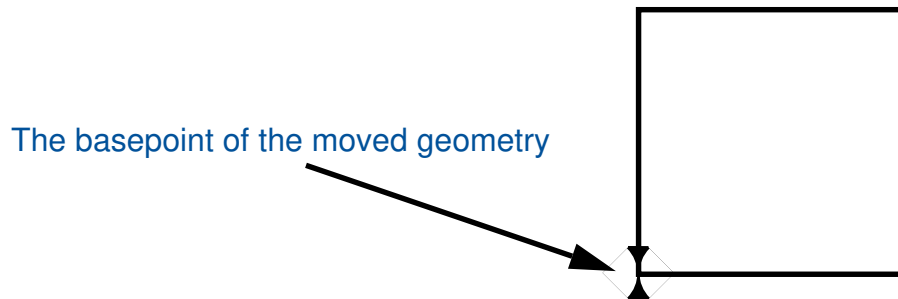


Figure 2-23. Moved Geometry, and its Basepoint

The moved geometry is automatically unselected, and you are prompted automatically to select another geometry to move. If you do not want to move another geometry, cancel the Select Area prompt bar that is displayed.

If you decide that you accidentally moved something to an incorrect location, you can undo the move by choosing the **[Shapes] > Geometric Undo** menu item.

Lab Exercise

This lab exercise gives you more experience with the basic commands used in Board Station LIBRARIAN for creating and manipulating geometry data.

Upon completion of this lab exercise, you can:

- Set up an appropriate grid and edit and viewing layers for the geometry you need to create.
- Create, edit, move, and copy graphics in either the absolute coordinate system or the delta coordinate system.
- Create lines, polygons, circles, and arcs.
- Use the Select Filter to control what kinds of objects can be selected for editing.
- Edit an existing geometry by adding fillets or trimming.
- Add or position graphics relative to the location of existing graphics by snapping to the existing geometry.

Turn to Module 3—Lab 2: "Basic Geometry Creation Techniques".

Lab 2

Basic Geometry Creation Techniques

Introduction

In this lab exercise you explore the techniques for creating geometries. Upon completion of this lab exercise, you can:

- Set up an appropriate grid and the edit and viewing layers for geometries you need to create.
- Create, edit, move, and copy a geometry in either the absolute coordinate system or the delta coordinate system.
- Create lines, polygons, circles, and arcs.
- Use the Select Filter to control what kinds of objects can be selected for editing.
- Edit an existing geometry by adding fillets or trimming.
- Add or position a geometry relative to the location of an existing geometry by snapping to the existing geometry.

Procedure

In this lab you create a geometry to gain experience using the geometry creation techniques you need to create your board and other geometries. The experimental geometry you create has features similar to the final board you make later. When you finish creating the experimental geometry, you delete it. Although you are not using the experimental geometry in your design, you are using the techniques learned to create other geometries, such as the board. The first thing you learn is another way to make lines, and trim intersecting lines to make a corner.

Preparation for Lab

First, you invoke the Design Manager and LIBRARIAN.

1. Invoke the Design Manager by entering the following in a shell:

```
$MGC_HOME/bin/dmgr
```

2. Find the LIBRARIAN icon in the Tools window, as shown in Figure 2-24. Invoke LIBRARIAN by placing the cursor on the LIBRARIAN icon and double-clicking the Select mouse button.



Figure 2-24. LIBRARIAN Icon

3. In the Specify Invocation Mode dialog box that appears, choose **Invocation Mode: Stand Alone**. Then press the **OK** button in the dialog box.

A Report-Startup message might appear in the middle of the LIBRARIAN Session window. This report is a list of notes concerning the files used to invoke the LIBRARIAN tool.

4. After reading the report notes, close the report window, and then maximize the size of the LIBRARIAN session window to fill the display.

Creating a Generic Geometry

1. Choose the **Geometries > Create Geometry > Generic** menu item. In the dialog box, enter the name **experiment**, and **OK** the dialog box.

You use this generic geometry edit window to create your experimental geometry.

2. Choose the **Setup > Edit Layer** menu item. In the Set Edit Layer dialog box, place the cursor on Component Body Outline in the list, and then choose OK.

The name COMPONENT_BODY_OUTLINE is displayed in the upper-right corner of the edit window, in the color assigned to data created on that layer. For this lab exercise, the edit layer you use is not important, but this layer is a color that is easy to see. If you want to use another edit layer, you can.

3. Choose the **Setup > Line width** menu item. In the prompt bar, enter 0.0 and click OK.
4. Choose the **Setup > Grid** menu item. In the Set Grid dialog box, set the X increment to **0.05** and the display interval of **2**.

The Y increment is set automatically to the same as the X increment, unless you enter another number for the Y increment. Because the display interval is 2, you see grid dots every 0.1 inch, but the cursor and geometry snap (when snapping is on) to 0.05 grid increments.

5. Zoom out to see a larger area by either clicking twice on the *O* (zoom out) icon in the upper-right corner of the edit window, as shown in Figure 2-25, or by placing the cursor in the edit window, and typing **zoom out 4** (when you type, a popup command line appears), and pressing the Return key.

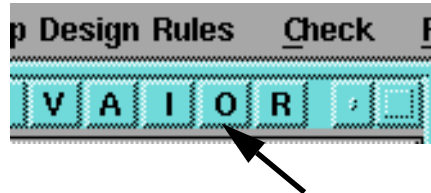


Figure 2-25. The Edit Window Zoom Out Icon

If the grid points are not visible, zoom in (click on *I* icon, or type **zoom in 1**) until the grid points are visible. If you move the cursor from the extreme left edge of the edit window and note the absolute x,y coordinate, and then move the cursor to the extreme right edge of the edit window and note the absolute x,y coordinate, you can verify that about 6 inches of horizontal area are visible.

6. Choose the [Top Menu] Shapes menu item.

The Top Menu is removed, and the Shapes menu pops up so you can make a selection from it. From now on, the Shapes menu is the top-level popup menu. This saves you the time of having to cascade down through the Top Menu all the time. The Shapes, Dimension, Drawing, Text, and Attributes menus can all be defined to be the top-level popup menu just by choosing them from the Top Menu. If you want to have access to the Top menu again, choose Return to Top Menu from the Shapes or other popup menu. Because you are going to be doing a lot of work in the Shapes menu, you want to have the Shapes menu as the top-level popup menu now.

You will be creating the geometry shown in Figure 2-26.

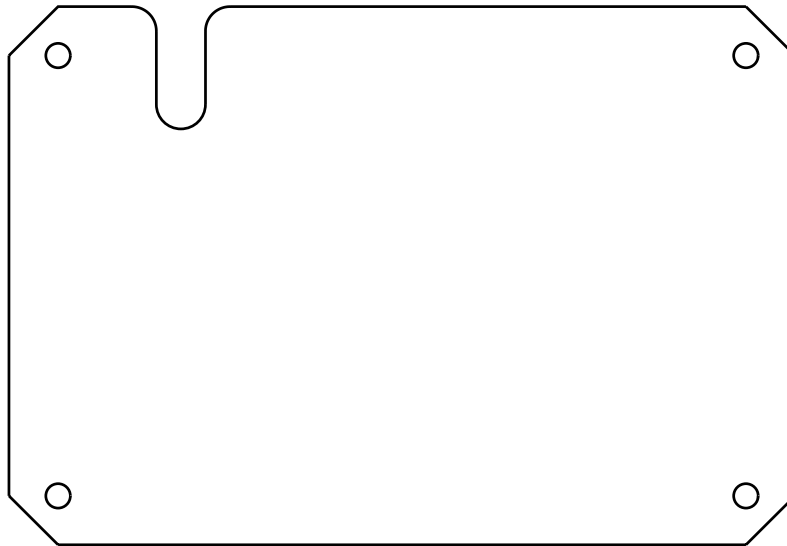


Figure 2-26. Experimental Board

7. Choose the **[Shapes] Add Line > Horizontal** menu item. When the prompt bar is displayed, and you are prompted for the Anchor Point, place the cursor on the geometry origin (Absolute coordinate $x=0, y=0$), and click the Select mouse button.

A horizontal line is created that is as long as the viewed area is wide. The prompt bar returns, and you are prompted for another Anchor Point.

8. Place the cursor 2 inches above the first horizontal line, and click the Select mouse button again. Cancel the prompt bar when it returns.

You now have two parallel horizontal lines. Next, you will add two vertical parallel lines.

9. Choose the **[Shapes] Add Line > Vertical** menu item. When the prompt bar is displayed, and you are prompted for the Anchor Point, place the cursor about 1.5 to 2 inches to the left of the geometry origin (about midway between the center of the viewed area and the left edge), and click the Select mouse button.

10. When the prompt bar returns, place another vertical line midway between the geometry origin and the right edge of the viewed area, so the four lines create a rectangle between their intersections. Cancel the prompt bar.

You now have four lines, as shown in Figure 2-27.

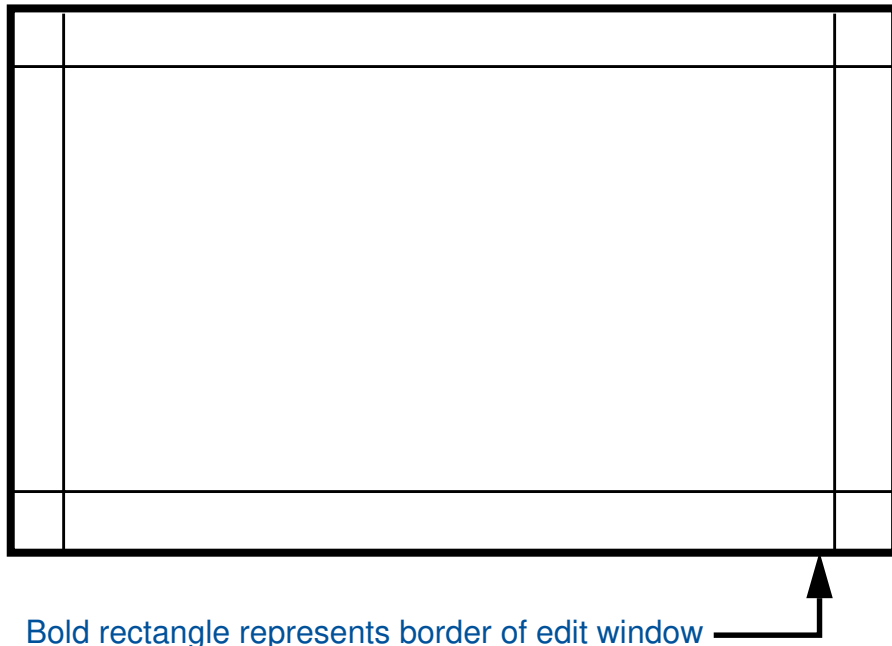


Figure 2-27. First Four Lines

Creating a Chamfer

Next you are going to chamfer and clip (trim) the corners to make a clean rectangle with chamfered corners.

1. Choose the [**Shapes**] **Add Line > Chamfer** menu item. In the dialog box, Tab to the Distance prompt, and enter **0.2**. Press the Tab key three times to highlight the Clip Option prompt and click on the prompt arrows until **clip_both** displays. Press the Tab key (or the Shift-Tab key) until the From prompt is highlighted, then place the cursor on the left vertical line and press the Select mouse button. When you are prompted for the To point, place the cursor on the lower horizontal line and press the Select mouse button. Refer to Figure 2-28. When you are done, Cancel the prompt bar.

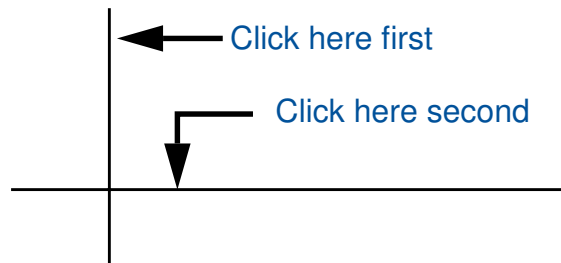


Figure 2-28. Creating a Chamfer at Lower-Left Corner

A 45 degree chamfer is created in the lower-left corner as shown in Figure 2-29. When creating chamfers with the **clip_both** option, always click on the portion of the line segments you want to keep. The line segments on the other side of the intersection are removed.

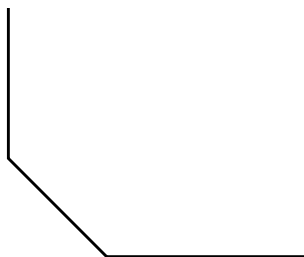


Figure 2-29. The Complete Chamfer

2. Create chamfers for all the other three corners of the rectangle.

Adding a Slot

Next you add a slot that cuts vertically down from the top edge of the rectangle and has rounded ends.

1. Choose the **[Shapes] Add Line > Slot** menu item. In the dialog box, click on the **Options** button. In the options dialog box, enter a diameter of 0.2, and choose Arcs on Both Ends. **OK** the dialog box.
2. Place the cursor about 0.5 to 0.75 inch above the top horizontal line of the rectangle, about 0.75 to 1.0 inch to the right of the left side of the rectangle, and click the Select mouse button. Refer to Figure 2-30.

X ← First point about here



Figure 2-30. The First Point of the Slot

This marks the first point of the slot, where the center-point of the arc will be. You are now prompted for the second point of the slot.

3. Move the cursor down (you see a *rubberband* line showing the distance between the center-points of the two arcs of the slot) so that the slot extends down into the rectangle about 0.5 inch from the top of the rectangle to the center-point of the lower arc of the slot. Click the Select mouse button. When the slot is drawn, Cancel the prompt bar. Refer to Figures 2-31 and 2-32.

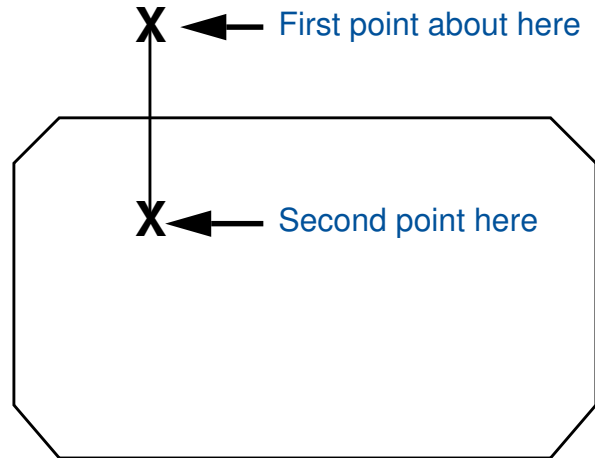


Figure 2-31. The Second Point of the Slot

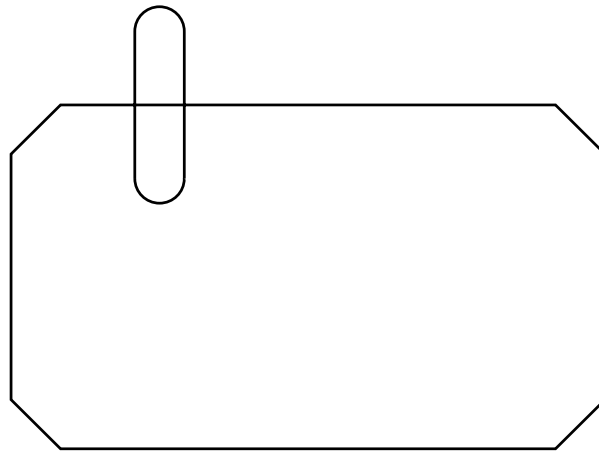


Figure 2-32. The Roughed-in Slot

Next you will trim the vertical lines of the slot to meet the horizontal line of the rectangle.

4. Choose the **[Shapes] Trim** menu item. When the prompt bar displays, place the cursor on the left vertical line of the slot, above the horizontal board outline, and click the Select mouse button. Next, place the cursor on the horizontal board outline, and click the Select mouse button again. Refer to Figure 2-33.

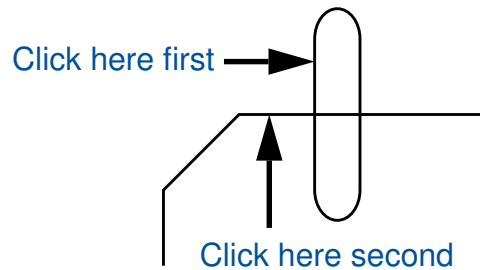


Figure 2-33. Trimming One Line of the Slot

The line you click on first gets trimmed to the line you click on second. You can either trim a line that extends beyond another line back to make a corner, or you can trim a line to make it extend until it contacts another line.

5. Trim the right vertical line of the slot down to meet the horizontal line of the rectangle. You need to indicate your first point as being the upper half of the vertical line of the slot, and the second point as the horizontal line of the rectangle, as shown in Figure 2-34.

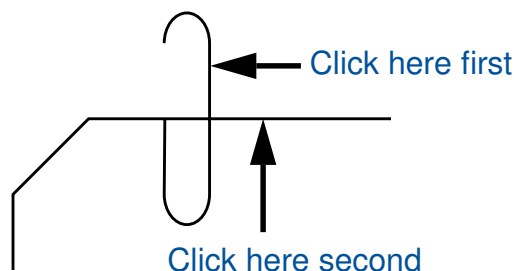


Figure 2-34. Trimming the Second Line of the Slot

Trim the lines as shown in Figure 2-35.

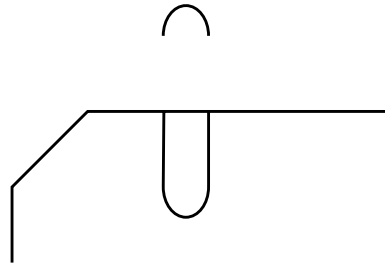


Figure 2-35. Trimming the Sides of the Slot

If you specify the first point as the lower half of the vertical line of the slot, the line would be trimmed up, leaving the upper half of the slot in place. You want to have a slot into the rectangle (which you can think of as the board), not a projection out of the board boundary. You now have an arc hanging off in space just above the rectangle's edge. Next, you delete this arc.

6. Cancel any prompt bars.
7. Choose the **[Shapes] Select > Select Area** menu item, and then choose **Options** from the Select Area prompt bar. In the options dialog box, choose Arcs, and **OK** the dialog box. Use the Select mouse button to select the arc by placing the cursor below and to the left of the arc, pressing and holding down the Select mouse button, and then moving the cursor so the displayed select area box encloses only the arc. Finally, release the Select mouse button. Check the *Selected*: count in the status window to be sure only one item is selected. Cancel the select area prompt bar, if it repeats.

Only the arc is selected. To make sure that only a certain type of geometry is selected, use the select area function (or menu item) options dialog box. Selecting a geometry type is especially important when you have several kinds of geometry data very close to each other, and you want to select only one of them. You can also use this feature to select all the text on a geometry without selecting anything else.

8. Place the cursor in the edit window, and type **del**. When you have entered the del command (delete) in the popup-command line, press the RETURN key.

A dialog box is displayed, and you are prompted for the type of object to delete.

9. In the dialog box, choose **other**, and then **OK** the dialog box.

The selected arc is deleted.

Next you divide the horizontal line at a point between the top ends of the slot. You do this so the two resulting horizontal line segments can each be trimmed back to the top edges of the slot later.

10. Choose the [Shapes] **Divide > Divide Point** menu item. When the select area prompt bar displays, place the cursor on the upper horizontal line of the rectangle, and click the Select mouse button (one item is selected). When the divide point prompt bar displays, and you are prompted for a point, place the cursor on the horizontal line you just selected between the two sides of the slot, and click the Select mouse button. Refer to Figure 2-36.

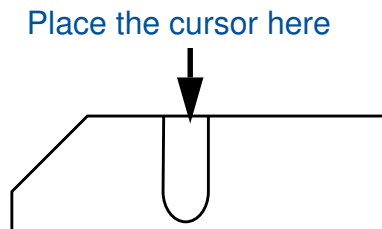


Figure 2-36. Specifying the Dividing Point

You now have two items selected; and the line is divided.

11. Unselect all items by pressing the Unselect All function key.

This is the most convenient way to unselect all items. Next you add fillets at the top edges of the slot. When you do, the ends of the horizontal line segments are automatically trimmed.

12. Choose the **[Shapes] Add Arc > Make Fillet** menu item. Enter 0.1 for the radius, then press the TAB key to be prompted for the *From* point. Make sure the Clip Option is set to **clip_both**. Place the cursor on the horizontal line near the top of the left side of the slot, and click the Select mouse key. When you are prompted for the *To* point, place the cursor on the vertical line of the left edge of the slot, and click the Select mouse button again. Refer to Figure 2-37.

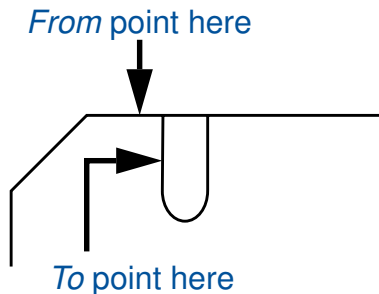


Figure 2-37. Making the First Fillet

The fillet is formed as shown in Figure 2-38. It does not matter what order you choose the lines when making a fillet. You could have picked the vertical line first, and then the horizontal one.

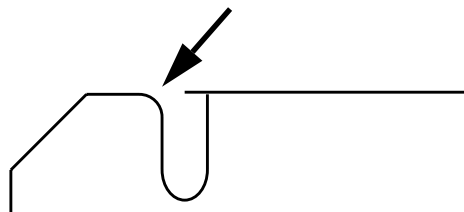


Figure 2-38. The First Fillet

13. Make another fillet for the upper-right corner of the slot. Cancel the prompt bar.

Adding Mounting Holes

Next you add mounting holes to your practice board. You place the circles relative to an existing geometry. In this case, you place the circles relative to where the corners of the board were before you added the chamfers. First, you place a basepoint at the location where the intersection was, and then use the Delta coordinate system to place the circle some distance from the basepoint.

The Delta coordinate system uses the basepoint for its origin, and it is independent of the Absolute coordinate system. Normally, the basepoint is automatically placed at the last point you specified when you add a geometry. However, you can specify to move the basepoint to any location in a coordinate system, or to any point on an existing geometry. In the example you will try next, you will *snap* the basepoint to the intersection of two lines. You can snap the basepoint to any type of geometry, such as the center of a circle, the midpoint or endpoint of a line, and so on. Using the snapping features of LIBRARIAN to position a geometry or the basepoint can be more accurate than clicking on a location with the Select mouse button, especially if your design has a very fine grid.

1. Choose the **[Shapes] Move > Basepoint** menu item. When you are prompted for a location *do not* click the Select mouse button. Read the following explanation before continuing.

You are prompted for a location. To provide the location, you are going to snap to the intersection of two lines. Because the first circle you add will be in the lower-left corner of the board outline, you will be snapping to the intersection of the left vertical board edge, and the bottom horizontal board edge. These lines do not really intersect anymore, because you placed a chamfer there, but you can still snap to their projected intersection.

2. Choose the **[Shapes] Snap > Intersection** menu item.

You are prompted for the location of two lines that intersect. The Snap Intersection prompt bar displays on top of the Move Basepoint prompt bar. This feature of using one prompt bar to satisfy prompts of another prompt bar is called Mid-Command Freedom. Frequently, you need to use a second menu item to provide information to a previous prompt bar.

3. Place the cursor on the left vertical edge of the board, and click the Select mouse button.

The vertical line highlights. You are prompted for another point.

4. Place the cursor on the lower horizontal edge of the board, and click the Select mouse button.

The horizontal line highlights briefly, then the basepoint is placed at the projected intersection of the two lines as shown in Figure 2-39. Finally, both lines are unhighlighted. Now add a circle and position it relative to the basepoint's location.

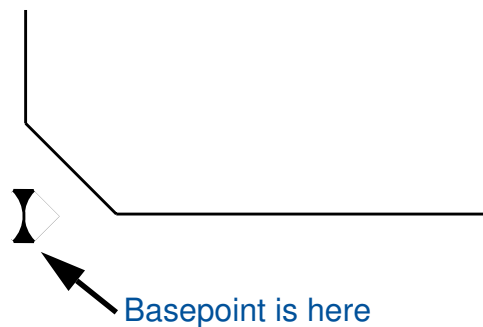


Figure 2-39. The Basepoint

No prompt bars are visible now. When you used the Snap Intersection prompt bar to satisfy the Move Basepoint prompt bar, both are removed, because both had enough information to complete their jobs.

5. Choose the **[Shapes] Add Circle > Center Value** menu item. In the prompt bar, enter **0.1** for the Radius, and press the TAB key to highlight the Location (+) prompt.

To provide the location, you will use the Delta coordinate system, and provide a X and Y coordinate distance from the basepoint.

This is another case where you use Mid-Command Freedom to use one prompt bar to satisfy the needs of another.

6. Choose the **[Shapes] Snap > Delta** menu item. In the dialog box, enter Delta X = 0.3, and Delta Y = 0.3. Press the TAB key to highlight the From prompt. Place the cursor on either the up or down arrow in the From prompt, and click the Select mouse button until the word *basepoint* is displayed in the prompt. **OK** the prompt bar. Cancel the repeating Add Circle prompt bar.

After you **OK** the prompt bar, you might have to move the cursor back into the edit window before the circle you added is displayed, as shown in Figure 2-40. If there are any prompt bars visible, cancel them.

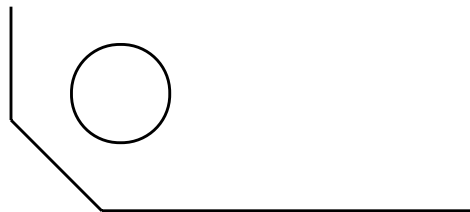


Figure 2-40. The Circle

7. Repeat the steps to add circles in the other four corners of the board.

The Delta X and Y values you give can have the same dimensional value, but need to be negative in some cases to correctly position the circle.

This process of placing a basepoint relative to a piece of geometry data and then placing new geometry relative to the basepoint is used frequently. Practice this procedure until you are comfortable with it.

If you don't have an existing geometry, you can still use the Delta coordinate system to create a geometry. In the next procedure, you will create a simple rectangle by placing the first point in an Absolute coordinate location, and then placing the other points using the Delta coordinate system. In this way, you can specify the location of the next point based on the location of the previous point.

Deleting Geometries

First you select and delete all the geometries you have just created. Remember, these geometries was just for practice.

1. Choose the **Setup > Select Filter** menu item. In the dialog box that is displayed, choose Set All. **OK** the dialog box.

The select filter defines what the Select mouse button and the Select Area and Select All menu items and Function keys can select. The select filter stays in effect until you change it during the current session.

Another way to define what the Select Area and Select All menu items can select is to use the Options dialog box for those menu items, which looks the same as the select filter. This method defines what can be selected only once. If you do not use the Options dialog box to determine what can be selected, the system uses what is defined in the select filter.

It is important to remember what the select filter does, and how it works. You will probably use the select filter and the Select Area and Select All Options dialog boxes frequently.

Now anything can be selected for the remainder of this session, because that is what you defined in the select filter.

2. Choose the **[Shapes] Select > Select All** menu item.

Because you defined the select filter to select anything, all items are selected.

3. Place the cursor in the edit window, type **del**, and press the Return key.

When you type, a popup command line is displayed. You entered the short-hand command name for delete. You are going to delete the experimental geometry you just finished creating. You will use all the techniques you have learned so far to create the real board later in this lab.

A verification dialog box is displayed.

4. In the dialog box, choose **Other**, and then **OK** the dialog box.

Using the Delta Coordinate System

Now that you have a clean area in which to work, you will create a rectangle using the Add Line menu item, and the Delta coordinate system.

1. Choose the **[Shapes] Add Line** menu item.

In the dialog box, notice that the Points prompt shows a series of small + icons stacked up. This means you can enter many points, and the resulting line will have a vertex at each of those points. When you see only one + sign at a location prompt, it means you can only enter one point, such as when you specify the center of a circle.

2. Choose the **[Shapes] Snap > Absolute** menu item. Enter $x=0, y=0$ in the prompt bar, and press the Return key to **OK** the prompt bar.

You can use either Return or **OK** to complete dialog boxes and prompt bars.

You have specified the first vertex of the line. If you move the cursor in the edit window, you will see a *drag image* of the line, with one end fixed at the location you specified, and the other end moving with the mouse. Next, you will specify a point relative to the last point you specified.

The Add Line prompt bar, prompts you to enter another point in the line.

3. Choose the **[Shapes] Snap > Delta** menu item. In the prompt bar, enter $x=3, y=0$. In the From prompt, click on the scroll arrows until *lastpoint* is displayed, and press the Return key to complete the prompt bar.

You now have a horizontal line segment 3 inches long (inches are the current user unit of measure, by default). A drag image moves between the second vertex, and the cursor.

4. Place the cursor in the edit window, and hold down the Shift key. While holding down the Shift key, gently click the Menu mouse button without moving the cursor.

The **[Shapes] Snap > Delta** menu item repeats. The system remembers the previous menu item used from each of the pulldown menus and the popup menu. If you place the cursor in the area of any menu (on the menu name in the menu bar for a pulldown, or in the edit window for a popup), hold down the Shift key and click the Menu mouse button without moving the mouse, the previous menu item from that menu is repeated.

If the Snap Delta prompt bar did not repeat, you might have accidentally moved the mouse when you tried to repeat the menu item. If it did not repeat correctly, make sure the only the Add Line prompt bar is visible, and then choose the **[Shapes] Snap > Delta** menu item in the usual way.

5. In the Snap Delta prompt bar, enter $x=0$, $y=2$, and **OK** the prompt bar. To view the resulting lines, you might have to move the cursor.

Two sides of the box are now formed. The Add Line prompt bar is visible; there are no other prompt bars.

6. Repeat the Snap Delta prompt bar again, and enter $x=-3$, $y=0$. **OK** the prompt bar.
7. Repeat the Snap Delta prompt bar again, and enter $x=0$, $y=-2$. **OK** the prompt bar. **OK** the Add Line prompt bar. Cancel the Add Line prompt bar that repeats.

The box is completed. You see there are many ways to create geometries. Use the methods that are most productive for you.

8. Select all the geometries, and delete them.

You can use the [Shapes] Select > Select All menu item, or use the Select mouse button. You use the delete command (del) to delete selected geometries.

As the labs progress, you will be given less and less detail in the instruction for procedures you have already done. From now on, you will be given less detail for viewing, creating, selecting, and deleting geometries, unless the specific action you are required to do is new to you.

9. Close the generic geometry edit window *experiment*.

You no longer need this edit window, as it was for experimentation only. In the next procedure, you will create the board geometry you will use during the remaining lab sessions of this training.

10. Close LIBRARIAN, and log off the workstation.

Congratulations! You have completed Lab 2: "Basic Geometry Creation Techniques". The next lesson is Lesson 3: "Creating Component Geometries".

Lesson 3

Creating Component Geometries

Now that you have the skills to create graphics, you learn to use those skills to create geometries for components. Later, you use those components in a design.

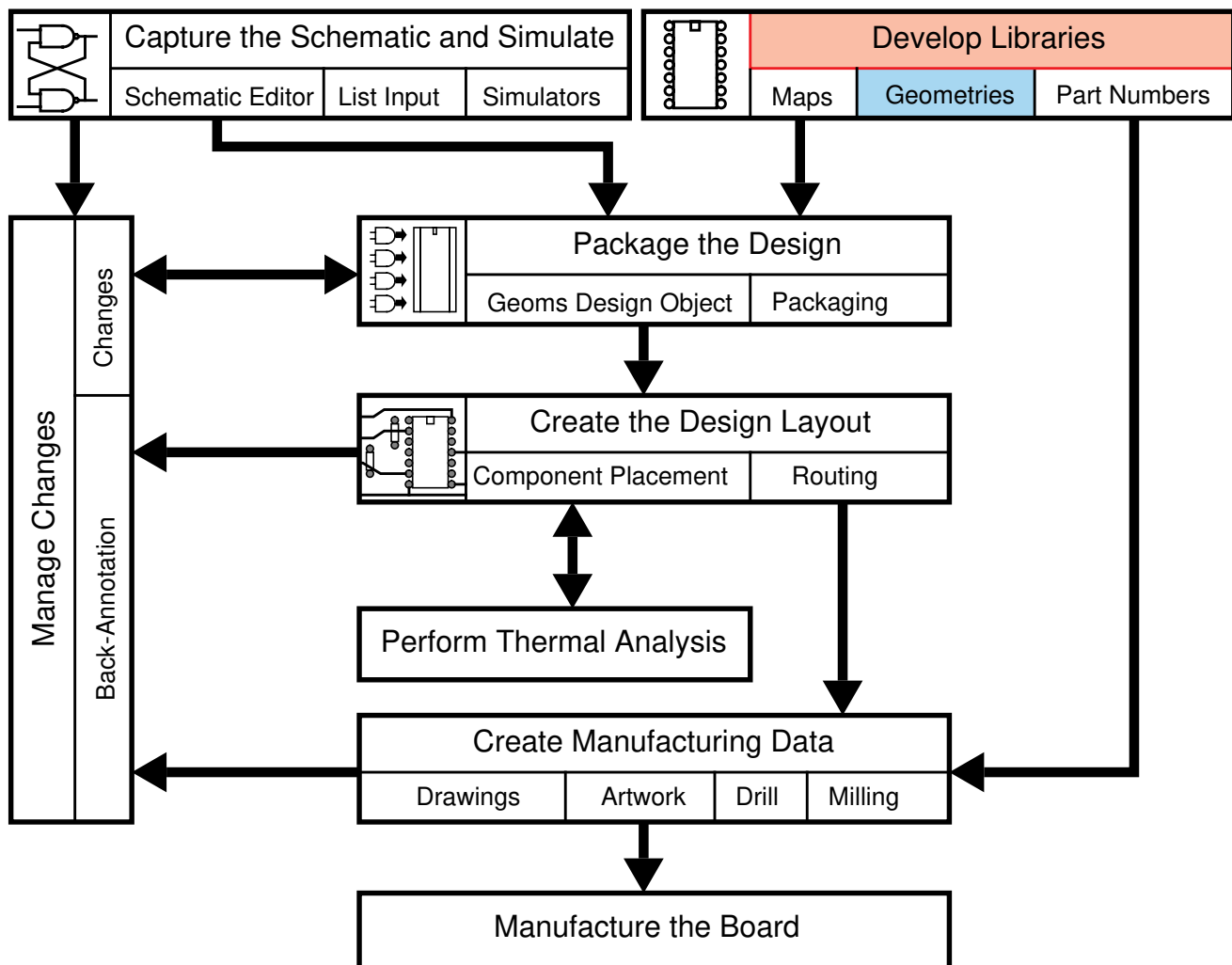


Figure 3-1. The PCB Design Process

Objectives

In this lesson, you complete your study of how to create component and padstack geometry types by learning about attributes. A specific set of attributes is associated with each geometry type. At the end of this lesson, you can:

- Describe the process for creating via, padstack, component, and generic geometries.
- Identify the attributes for each geometry type, and describe the purpose of each attribute.
- Describe how to add pins to a component geometry.
- Describe how to check and save geometries.

Creating New Geometries

The general process for creating geometries is to:

1. Specify the type of geometry you want to create by choosing a geometry type from the **Geometries > Create Geometry** pulldown menu, as shown in Figure 3-2.

From this menu you can choose to create components, boards, pins, vias, and so on. When you choose the type of geometry you want to create, LIBRARIAN knows what attributes to add, and will prompt you for values from an appropriate dialog box.

An attribute is information added to the geometry either for other Board Station applications, human readability, or for back-annotation into Design Architect for the schematic.

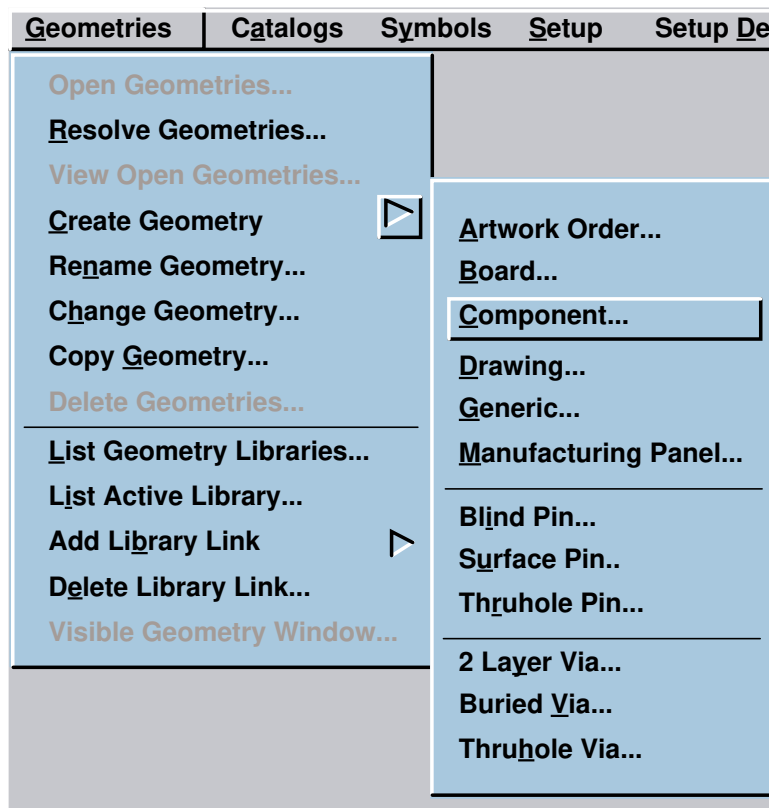


Figure 3-2. Geometries Pulldown Menu

2. In the dialog box that displays, enter the values of the attributes.
No attributes are required for a generic geometry, so no dialog box displays.

3. If you are creating board, component, drawing, generic, or manufacturing panel geometry, add the graphics (lines, arcs, and so on) that you want.

Graphics are automatically created for pins and vias, based on information you supply in their dialog boxes from step 2. Artwork Orders require no graphics.

4. Check and then save your geometry.

You can copy and alter existing geometries to create new geometries. The procedures that describe changing a geometry apply when you are altering an existing geometry and when you are modifying a copy of an existing geometry to create a new geometry.

Changing Generic Geometries

In previous lessons and labs of this module, you learned how to create new generic geometries. Generic geometries require no attributes.

You can change an open generic geometry into other geometry types. To open a pre-existing generic geometry and change its type, you:

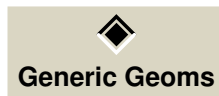
1. Invoke LIBRARIAN on a design. When invoking from the Design Manager, you double-click the Select mouse button on the Librarian icon. Then in the dialog box that displays, you choose *On a Design*. When invoking from a shell, you use the following syntax: **librarian design_pathname**.

You must invoke LIBRARIAN on a design so you can open the geometries that are within the design and change them.

2. Open a geometry by choosing the **Geometries > Open Geometries** menu item from the menu bar.

The Open Geometries dialog box displays. It contains a list of all the geometries included in the design. The type (generic, board, pin pad, via pad, and so on) of each geometry is listed next to each geometry name. In this way, you can identify which components are generic.

The **Geometries > Open Geometries** menu item is not available if you do not invoke LIBRARIAN on a design.



3. Near the bottom of the dialog box, select the **Generic Geoms** button to filter the list of geometries to generic geometries only.
4. In the dialog box, you place the cursor on the name of any generic geometry, click the Select area mouse button, and then choose OK to close the dialog box and open an edit window with the specified geometry.

5. Place the cursor in the edit window containing the generic geometry you just opened, or any other generic geometry, and hold down the Menu mouse button.

The popup menu will appear as shown in Figure 3-3. The **Change This Geometry** submenu of the popup menu allows you to change a generic geometry into a board, component, pin padstack, via padstack, artwork order, drawing, or panel geometry type.

The **[Top Menu] Change This Geometry** submenu items change the attributes of generic geometry so LIBRARIAN recognizes it as a board, component, or whatever geometry type you specify.

If you choose **[Top Menu] Change This Geometry > Board**, the generic geometry changes to a board geometry. However, it will not yet have physical layers and other board attributes defined.

After you choose **[Top Menu] Change This Geometry > Board**, and the geometry type changes, you need to choose the **[Top Menu] Change This Geometry > Change This Geometry** menu item. This opens the Change Geometry Board dialog box, in which you specify the physical layers and other attributes of the board. This dialog box looks exactly like the Create Board dialog box you will study and use in the next lesson.

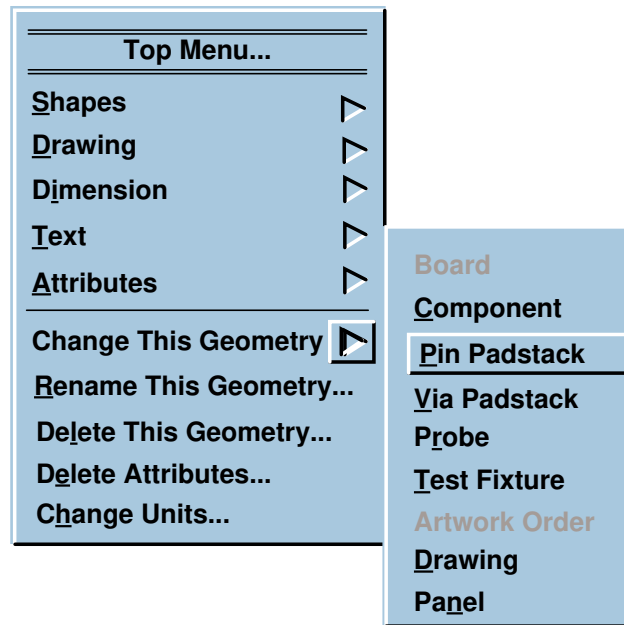


Figure 3-3. Change This Geometry Popup Menu

If another edit window is active, such as one that contains the board geometry, the contents of the popup menu might differ from what is shown here. The contents of the popup menu and its submenus are context sensitive, meaning that its contents change depending on the type of geometry you are working on.

Attributes

Attributes are basic information attached to a geometry, similar to properties in schematic capture. Required and optional attributes are automatically added to a geometry (except generic geometries) when you create a geometry with the userware dialog boxes. Most specifications that you set in the dialog boxes cause the appropriate attribute to be added to the geometry. The **Attributes** submenu of the popup menu provides items that let you change graphical attributes of a geometry. Examples of attributes are shown in Figure 3-4.

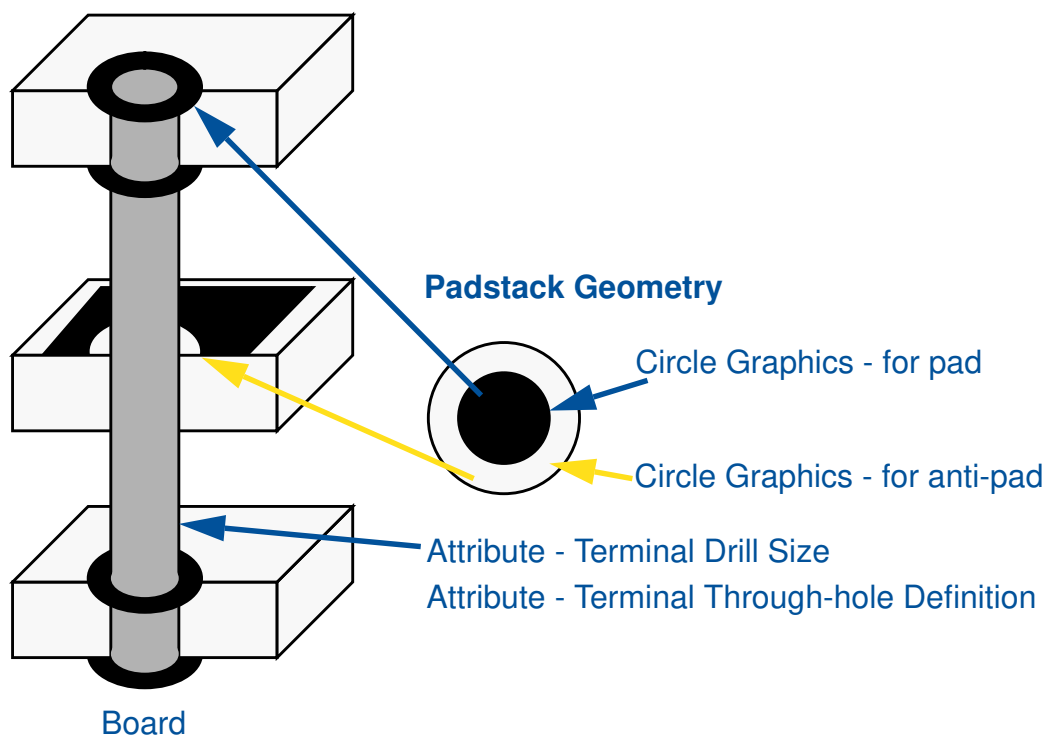


Figure 3-4. Attributes

Pin Padstack Attributes

The attributes associated with pin padstacks determine the type of pin padstack and the specification of a drill hole, if it exists. Choosing a pin padstack type from the **Geometries > Create Geometry** pulldown menu automatically adds the proper attribute type to the geometry.

- **Geometries > Create Geometry > Thruhole Pin** menu item adds the Terminal_thruhole_definition attribute.
- **Geometries > Create Geometry > Surface Pin** menu item adds the Terminal_surface_definition attribute.
- **Geometries > Create Geometry > Blind Pin** menu item adds the Terminal_blind_definition attribute.

Examples of these above pin padstacks are shown in Figure 3-5. When you add a drill hole to the padstack, a Terminal_drill_size attribute is added to the geometry.

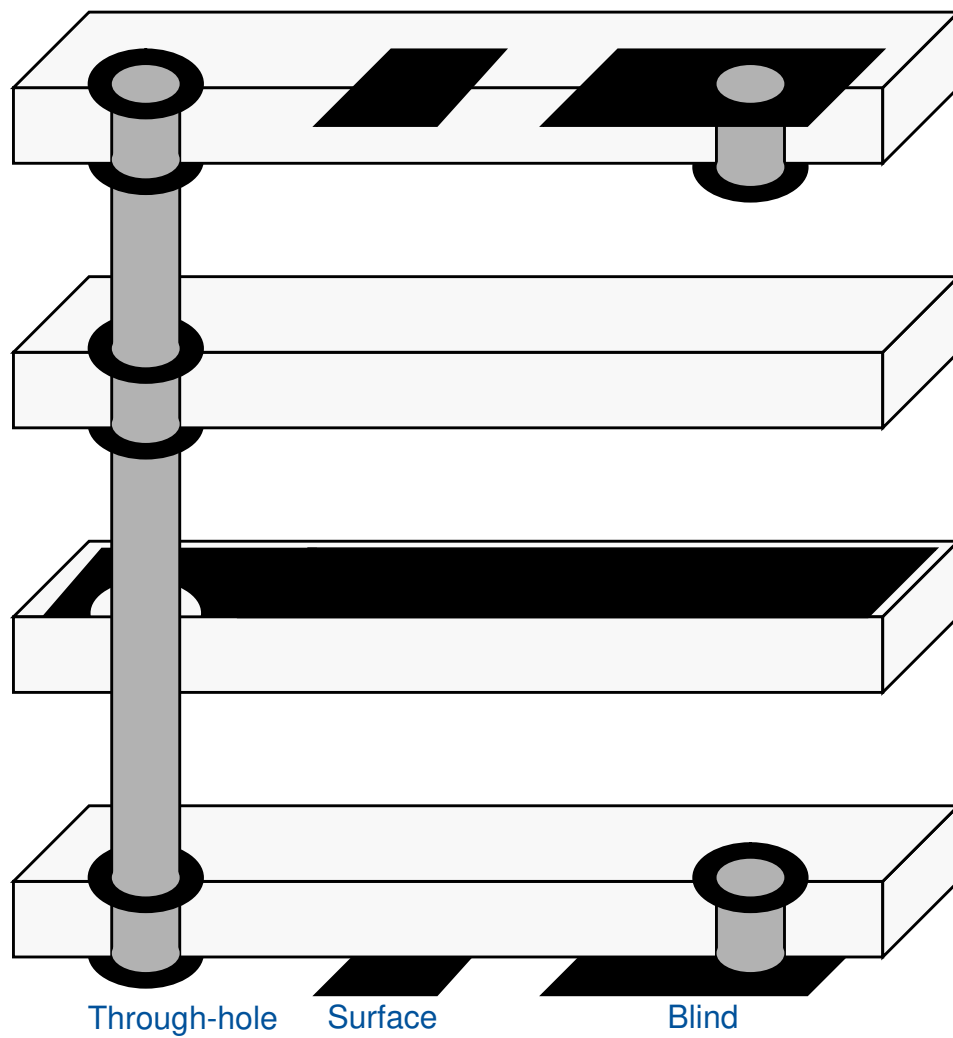


Figure 3-5. Pin Padstack Attributes

Via Padstack Attributes

The attributes associated with via padstacks also determine the type of via padstack and the specification of a drill hole, if it exists. Choosing a via padstack type from the **Geometries > Create Geometry** pulldown menu automatically adds the proper type attribute to the geometry. Examples of via padstacks are shown in Figure 3-6.

- **Geometries > Create Geometry > 2 Layer Via** menu item adds the Terminal_2 layer_definition attribute.

This type of via is not shown in the example. It is typically used for bonding pads in thick film hybrid applications.

- **Geometries > Create Geometry > Buried Via** menu item adds the Terminal_buried_definition attribute.

A buried via padstack is the most versatile type of padstack because a buried via can span any physical layers. You assign via rules to a buried via padstack to define the layers spanned by the via.

By defining via rules that span both surface layers of a board, a buried via is equivalent to a through-hole via.

By defining via rules that span one surface layer and an internal layer, a buried via is equivalent to a blind via.

By defining via rules that span only internal board layers, a buried via is equivalent to a true buried via. A single buried via can carry multiple via rules.

- **Geometries > Create Geometry > Thruhole Via** menu item adds the Terminal_thruvia_definition attribute.

When you add a drill hole to the padstack, a Terminal_drill_size attribute is added to the geometry.

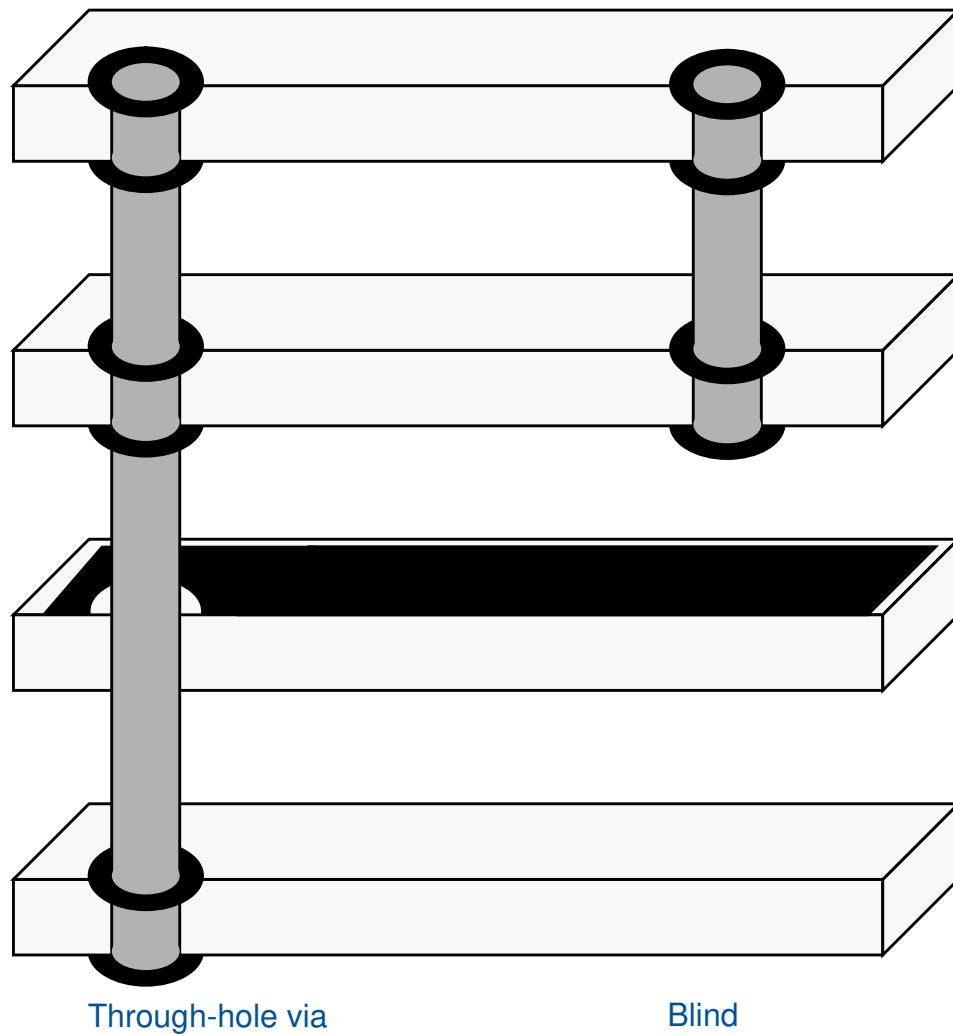


Figure 3-6. Via Padstack Attributes

Pin/Via Padstack Attribute

The `Aperture_shape_override` attribute associates an override shape with a particular layer in a pin or via padstack geometry. Mapping the shape to a laser photoplotter aperture allows you to flash a complex shape for the via or pin pad on the artwork film.



Split Power Plane

For split power planes, this attribute allows you to have a different shape on the artwork, depending on the voltage of the plane.

Component Attributes

Component geometries created in LIBRARIAN contain a set of required attributes. You can add optional attributes to the geometry to control special behavioral characteristics.

Using the **Geometries > Create Geometry > Component...** menu item displays the Create Component dialog box, as shown in Figure 3-7. You use this to add both required and optional attributes as you create the geometry.

You control the component characteristics with the following options in the Create Component dialog box:

- **Default Padstack Name**—adds the attribute `Component_default_padstack`, which associates a specific padstack geometry with the pins of this component.
- **Component Height**—adds the attribute `Component_height`. The combination of this attribute and `Component_thermal_outline` allows you to perform thermal analysis without generating AutoTherm parts.
- **Allow Component Outline to Overhang Board**—adds the attribute `Component_outline_overhang`, which allows a portion of the component outline to be outside the board outline.
- **Include Component In Bill of Materials**—when unselected adds the attribute `Component_not_in_bom`, which allows the component to be ignored when you create a bill of materials.
- **Allow Pins to be Movable**—adds the attribute `Component_pins_moveable`, which allows you to select and move a pin on the component to a new location in LAYOUT.
- **Edge Connector**—adds the attribute `Component_edge_connector`, which defines an area for an edge connector where other components cannot be placed.

Create Component			
Component Name <input type="text"/>		<input checked="" type="radio"/> Inches <input type="radio"/> CM <input type="radio"/> MM <input type="radio"/> Mils <input type="radio"/> Tenth Mils	
<input type="checkbox"/> Replace Existing Component		Units	
Component Pin Display Radius <input type="text" value="0.03"/>		<input type="radio"/> Inches <input type="radio"/> CM <input type="radio"/> MM <input type="radio"/> Mils <input type="radio"/> Tenth Mils	
Default Padstack Name <input type="text"/>		Component Height <input type="text"/>	
<input type="checkbox"/> Allow Component Outline to Overhand Board		<input type="checkbox"/> Allow Pins to be Moveable	
<input checked="" type="checkbox"/> Include Component in Bill of Materials		<input type="checkbox"/> Edge Connector	
Orientation <input type="radio"/> Orthogonal <input type="radio"/> Orthogonal & Dialog <input checked="" type="radio"/> All Angle			
<input type="radio"/> Top Surface <input type="radio"/> Bottom Surface <input checked="" type="radio"/> Both Surfaces	<input checked="" type="radio"/> Thru pin Component <input type="radio"/> Surface Component	Change Silkscreen Layer Mapping <input checked="" type="radio"/> None <input type="radio"/> Silkscreen_1 <input type="radio"/> Silkscreen_2 <input type="radio"/> Both	
Change Other Layer Mapping <input type="radio"/> Specify Layers <input checked="" type="radio"/> Ignore Attribute			
<input type="button" value="OK"/> <input type="button" value="Reset"/> <input type="button" value="Cancel"/> <input type="button" value="Help"/>			

Figure 3-7. Create Component Dialog Box

- Orientation Orthogonal/Orthogonal & Diagonal/All Angle**—adds the attribute Component_orthogonal_only or Component_diagonal_allowed. If you choose All Angle, no attribute is added. These attributes provide for limiting the rotation of the component.

- **Surface Top/Bottom/Both**—adds the attribute `Component_layout_surface` with the argument Top, Bottom, or Both. This attribute specifies the allowable placement surfaces.
- **Component Surface/Thru pin**—adds the attribute `Component_layout_type` with the argument Surface. This attribute defines the component as a surface mount component. The choice Thru pin is a default condition; no attribute is added.
- **Component Specific Layer Off**—adds the attribute `Component_specific_layer_off` with the argument Silkscreen_1 or Silkscreen_2. This attribute controls the display and artwork processing of graphic data on specific silkscreen layers.
- **Component Specific Layer On**—adds the attribute `Components_specific_layer_on` with the name of a specific layer as an argument. You add this attribute to a component geometry to enable the display of component data on a specific layer, regardless of the component placement.

You add other component attributes by making a component geometry edit window active, and then choosing one of the following menu items from the **Attributes** submenu of the popup menu, as shown in Figure 3-8:

- **Add Placement Outline**—adds the attribute `Component_placement_outline`, which defines the area covered by the component. This attribute is required.
- **Add Thermal Outline**—adds the attribute `Component_thermal_outline`, which can be used to generate wire frame models for thermal analysis for components without existing AutoTherm parts.
- **Add Routing Keepout Area**—adds the attribute `Routing_keepout`, which describes an area on the geometry where routing is restricted.
- **Add Trace Keepout Area**—adds the attribute `Trace_keepout`, which describes an area on the geometry where traces are not allowed. Vias can be added within a trace keepout area.
- **Add Via Keepout Area**—adds the attribute `Via_keepout`, which describes an area on the geometry where vias are not allowed.

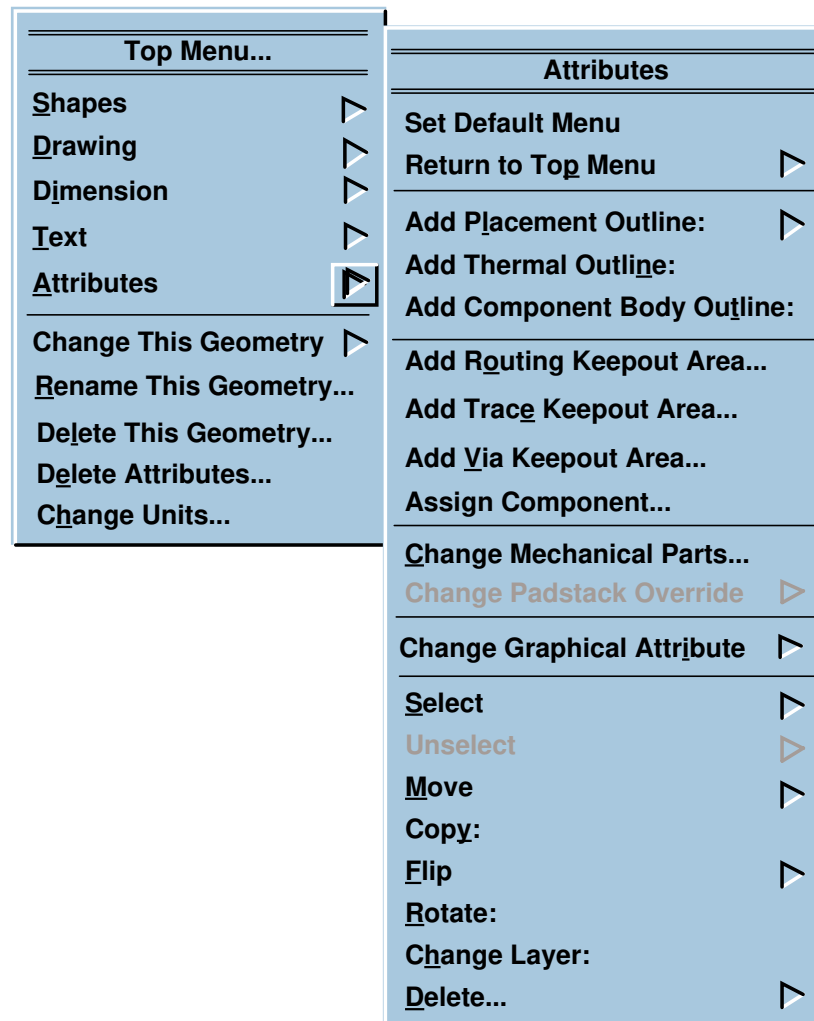


Figure 3-8. Attributes Sub-menu of the Popup Menu

- **Add Dimension Keepout Area**—adds a rectangular area to a geometry, either surrounding the geometry, or to one side of the geometry. The area prohibits the placement of horizontal and vertical dimensions within the area if the clearance_check optional switch of the \$add_horizontal_dimension() or \$add_vertical_dimension() functions is set when you add the dimension.
- **Assign Component Type**—assigns a type classification to a component. Used as a handle for assigning placement clearances to

components. Component type clearances override the default placement clearances specified by board geometry.

- **Change Mechanical Parts**—adds the attribute `Mechanical_parts`, which associates non-electrical part information with the component geometry. Mechanical parts include heat sinks, screws, lock washers, and card ejectors.
- **Change Padstack Override**—adds the attribute `Component_padstack_override`, which defines the padstack for a single pin on a component.

Adding Pins

For each pin you define, LIBRARIAN adds the `Component_pin_definition` attribute to the component geometry. As you add a pin to the component geometry, the Edit window displays the pad shape and pin number of the pin at the pin location.

When you are creating component geometries and no padstacks (board default, component default, or component override) are available in the LIBRARIAN session, the LIBRARIAN default is to draw a square pad shape for pin 1 and a round pad shape for all other pins. When you are creating component geometries and have padstacks available in the LIBRARIAN session, LIBRARIAN uses the padstack assignments to determine the pad shape to draw.

Several methods are available for adding pins to a component geometry:

- Adding pins individually.
- Adding pins in an array.
- Adding pins in a circular pattern.

The Pins sub-menu shown in Figure 3-9 is available when a component geometry Edit window is active.

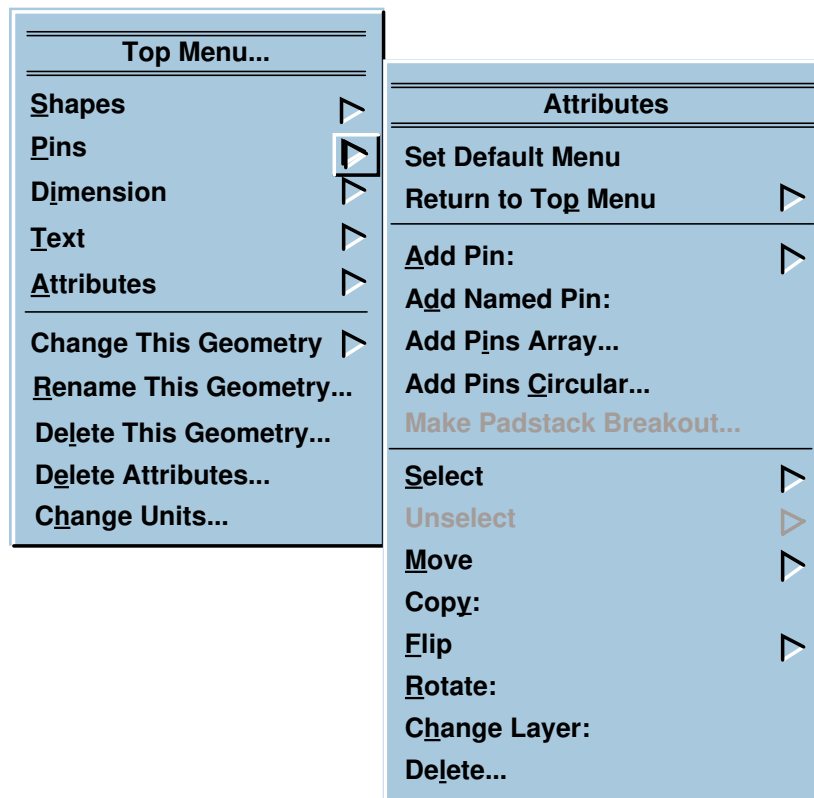


Figure 3-9. The Pins Sub-menu of the Popup Menu

Checking Geometries

You can apply the check to the active geometry only, to selected geometries in the current session, or to all geometries in the session.

- **Check > Geometry > Active Geometry**—checks the geometry in the active Edit window. LIBRARIAN checks the description of the active geometry for completeness and correctness. The check looks for conditions such as duplicate pins, redundant data, and missing attributes.
- **Check > Geometry > Geometries**—opens a dialog box with the option of checking all geometries in the current session or specified geometries by selecting geometries in the current session from a list box.

Lab Exercise

This lab exercise allows you to use some of the skills learned in the previous lab and apply them to creating pin and component geometries. Some of the geometries built in this lab are used in later sections.

Upon completion of this lab exercise you can:

- Create various types of padstacks.
- Create a variety of different component types.
- Save the new geometries in the Design library.

Turn to Module 3—Lab 3: "Creating Component Geometries".

Lab 3

Creating Component Geometries

Introduction

This lab exercise allows you to use some of the skills learned in the previous lab and apply them to creating pin and component geometries. You will use the geometries you build in this lab in later labs.

Upon completion of this lab exercise you can:

- Create various types of padstacks.
- Create a variety of different component types.
- Save the new geometries in your User library.

Procedure

In this session, you create some geometries that you will use later when you package components, place them on the board, and route traces. Some of these geometries are component geometries, some are the pin padstack geometries used for the components geometries, and some are via padstacks. The purpose is to give you a variety of experiences so you can better construct your own geometries.

Preparation for Lab

In this lab exercise, you use the LIBRARIAN tool to create additional geometries for your design.

First, you invoke the Design Manager and LIBRARIAN.

1. Invoke the Design Manager by entering the following in a shell:

```
$MGC_HOME/bin/dmgr
```

2. Find the LIBRARIAN icon in the Tools window, as shown in Figure 3-10. Invoke LIBRARIAN by placing the cursor on the LIBRARIAN icon and double clicking the Select mouse button.



Figure 3-10. LIBRARIAN Icon

3. In the Specify Invocation Mode dialog box that appears, choose **Invocation Mode: Stand Alone**. Press the **OK** button in the dialog box.

Another shell is created, which contains the transcript for LIBRARIAN. Soon, the LIBRARIAN session window is displayed. A Report-Startup message might appear in the middle of the LIBRARIAN Session window. This report is a list of notes concerning the files used to invoke the LIBRARIAN tool.

4. After reading the report notes, close the report window, and then maximize the size of the LIBRARIAN session window to fill the display.

Building the Pin Padstacks

During the next several procedures, you create three pin padstacks. First, you will build a round through-hole pin padstack geometry.

1. Choose the **Geometries > Create Geometry > Thruhole Pin...** menu item.
2. Fill in the dialog box as follows:

Pin Name: **th055028**
Drill Size: **.028**
Units: **Inches**

No PAD Shape

SIGNAL Shapes

Layer: **signal**

Circle

Diameter: **.055**

POWER Shapes

Layer: **power**

Circle

Diameter: **.080**

(Solder Mask) Single Shape

Circle

Diameter: **.065**

No OTHER Shapes

3. When the dialog box is complete, **OK** it.

When you OK the dialog box, a new edit window is created that contains the geometry for a round through-hole pin padstack.

Next, you build a square through-hole pin padstack geometry.

4. Choose the **Geometries > Create Geometry > Thruhole Pin...** menu item.

5. Fill in the dialog box as follows:

Pin Name: **th055028sq**
Drill Size: **.028**
Drill Hole Type: **Plated Hole**
Units: **Inches**

No PAD Shape

SIGNAL Shapes
Layer: **signal**
Circle
Diameter: **.055**
Layer: **signal_1**
Square
Width: **.055**

POWER Shapes
Layer: **power**
Circle
Diameter: **.080**

(Solder Mask) **Multiple Shapes**
SOLDER_MASK_1
Square
Width: **.065**
SOLDER_MASK_2
Circle
Diameter: **.065**

No OTHER Shapes

6. When the dialog box is complete, **OK** it.

When you complete the dialog box, another edit window is created on top of the previous edit window, and the new through-hole pin padstack geometry is created in the new edit window. The previous edit window, which includes the round through-hole pin padstack, is still open. To view it, you can choose the **Geometries > Visible Geometries Window... menu item**. A dialog box containing a list of all open windows is displayed. You can choose the name of the window you want to view, and when you **OK** the dialog box, the selected window is moved to the top of the window stack so you can view it.

Next, you build a surface pin padstack geometry.

7. Choose the **Geometries > Create Geometry > Surface Pin...** menu item.
8. Fill in the dialog box as follows:

Pin Name: **s70x30**

Units: **Inches**

(PAD) Single Shape

Rectangle

Width: **.070** Height: **.030**

(Solder Mask) Single Shape

Rectangle

Width: **.080** Height: **.040**

No PASTE_MASK Shapes

No OTHER Shapes

9. When the dialog box is complete, **OK** it.

Now you have three edit windows stacked one on top of the other in the session. Only one edit window is visible.

Building Via Padstack Geometries

During this procedure, you create a via padstack that will be used when you route traces on the board. You will build a round buried via padstack geometry.

1. Choose the **Geometries > Create Geometry > Buried Via...** menu item.
2. Fill in the dialog box as follows:

Via Name: **via040015**

Drill Hole

Drill Size: **.015**

Units: **Inches**

No PAD Shape

SIGNAL Shapes

Layer: **signal**

Circle

Diameter: **.040**

POWER Shapes

Layer: **power**

Circle

Diameter: **.065**

(Solder Mask) **Single Shape**

Circle

Diameter: **.050**

No OTHER Shapes

3. When the dialog box is complete, **OK** it.

Later, you define the design rules that are associated with this via. The design rules specify which physical board layers this via is allowed to connect.

Building Components Geometries

The next several sections lead you through creating component geometries.

Creating the Resistor Geometry

Now you will create a resistor geometry (named *rc07*) that you will use later in your board design. Refer to Figure 3-11 for a completed picture of the *rc07* resistor. Notice the spacing of the grid dots. Use the example spacing as a guide to the component's size when you create your geometry.

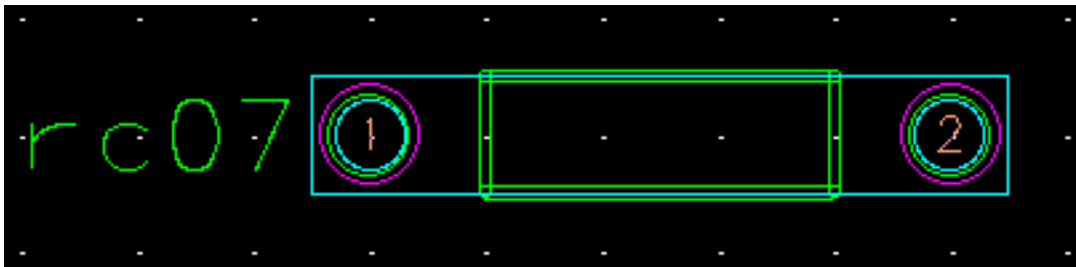


Figure 3-11. Completed Resistor Geometry

1. Choose the **Geometries > Create Geometry > Component...** menu item, then fill in the dialog box as follows. When you finish, **OK** the dialog box.

Component Name: **rc07**

Default Padstack Name: **th055028**

Leave all other information in the dialog box at the default values.

Another edit window is created in which you can create your geometry. Now set up a grid, add pins, and then add the body geometry.

2. Make sure the new edit window (with rc07 in its title) is active (meaning new edits can be added to it) by placing the cursor in the edit window and clicking the Stroke mouse button.

The window banner (where the window name is) changes color (to light blue by default). Only when an edit window's banner is highlighted is the edit window active. You can edit geometry in a window only when the window is active.

3. Choose the **Setup > Grid** menu item, and specify a grid of **.05** and a display interval of **2**. Choose the **Setup > Grid Snap On** menu item to turn grid snapping on.

If the Setup > Grid Snap On menu item is not in the Setup menu, and all you see is the Setup > Grid Snap Off menu item, it means the grid snapping is already on. This menu item toggles between on and off, depending on the current state of snapping.

4. Add the pin locations by choosing the **[Top Menu] Pins > Add Pin:** menu item.

A prompt bar displays. If you have trouble finding the menu item, remember that any menu name in brackets, [], means the menu is a popup menu found anywhere in the edit window.

5. Enter the following into the prompt bar, using the Tab key to move from prompt to prompt:

Numeric: **1** *Increment: 1* **Repeat**

You press the Tab key to highlight the next prompt in the prompt bar. As you repeatedly press the Tab key, the highlight moves from prompt to prompt across the prompt bar, and then starts over again at the first prompt.

The **Repeat** option is to the right of the location prompt. You might have to press the Tab key several times to see it. Because you chose the repeat option, the prompt bar reappears after you place each pin. This saves you time, because you do not have to choose the menu item again for each pin you add.

6. Press the Tab key to highlight the location prompt, which looks like a box with a small plus sign (+) in it.

Now the system is ready for you to tell it where to place the first pin. A location prompt with a single plus sign means only one location point is required. Sometimes you will see a location prompt with several small plus signs stacked on top of each other. When you see such a prompt, more than one location point is required, such as when you are creating a line or a polygon.

7. Place the cursor at the absolute coordinate x,y location (0.0,0.0), and then click the Select mouse button.

A pin is placed in the edit window, and the prompt bar is redisplayed, prompting you for another pin location.

8. Place the cursor 0.5 inches to the right of the first pin, at absolute coordinate, x,y location (0.5,0.0), and click the Select mouse button.

The pins identify the electrical connection points for this component and show the pin IDs. In this case the pin IDs are 1 and 2.

Because you defined the default padstack for this geometry to be the th055028 padstack when you opened this geometry (refer to step 1 of this procedure), the pins automatically appear with that padstack geometry at the pin locations.

9. Cancel the Add Pin prompt bar by placing the cursor on the Cancel button in the prompt bar and clicking the Select mouse button.

Next, you add a body outline to be silkscreened onto the board for manufacturing.

10. Choose the **Setup > Edit Layer** menu item. In the dialog box that is displayed, place the cursor on **SILKSCREEN**, and click the Select mouse button so Silkscreen is highlighted. Finally, **OK** the dialog box.

The active layer listed at the top-right of the screen has changed to **SILKSCREEN**, as shown in Figure 3-12. When you add geometry in this edit window, it is all added to the silkscreen logical layer, and the geometry is displayed in the color of that layer. By default, the color of the silkscreen layer is green.

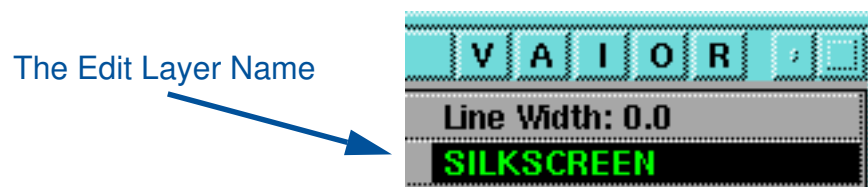


Figure 3-12. Location of the Edit Layer Name

In the next few steps, you add the body outline and placement outline as shown in Figure 3-13.

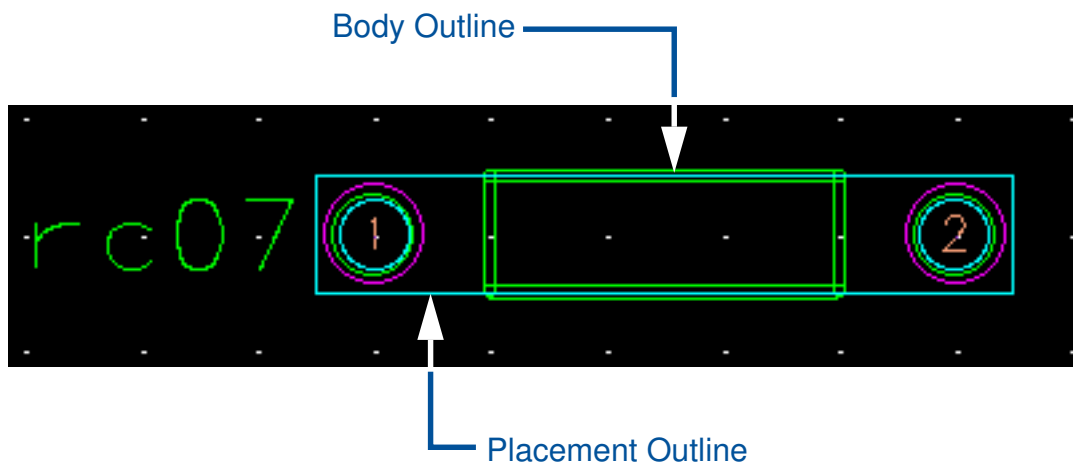


Figure 3-13. Resistor Body and Placement Outlines

11. Set the default line width to 0.01 inches by choosing the **Setup > Line Width:** menu item, and entering 0.01 in the prompt bar. Press the Return key, or place the cursor on **OK** and click the Select mouse button.

If, when you start typing, a popup command line is displayed (looks like a small gray one-line window), you accidentally moved the mouse after you chose the Setup > Line Width menu item, and before you entered the width. You need to remove this popup command line by pressing the Esc key (escape), then pressing the Tab key to let the system know you want to enter data into the prompt bar. You can see a small cursor flashing in the prompt bar, indicating that the system is ready for you to enter data there.

Anytime you accidentally see a popup command line, you can remove it using the Esc key. If you frequently see the popup command line when you try to enter data into a prompt bar, take sure that you do not move the mouse after you choose the menu item. You can also set the mouse sensitivity lower, so that greater mouse movement is required. See your system administrator for help adjusting the mouse sensitivity. Next, you draw the resistor body outline.

12. Choose the [Top Menu] **Shapes > Add Line > Add Line:** menu item.

A prompt bar displays. Create four line segments placed in relation to grid dots as shown in Figure 3-14.

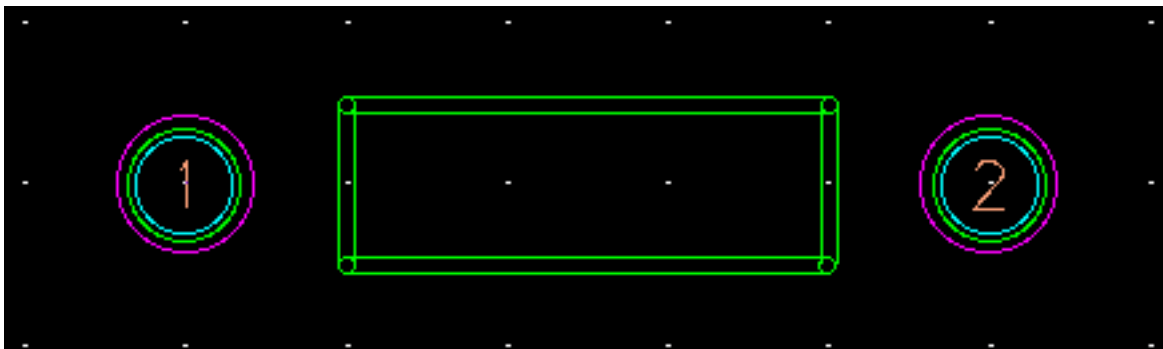


Figure 3-14. Resistor Body Outline

13. Place the cursor at the first corner of the outline, and click the Select mouse button. Move the cursor to the second corner, and click the Select mouse button again. Click the Select mouse button at the final two corners of the outline, then **OK** the prompt bar. **Cancel** the prompt bar when it repeats.

Instead of pressing OK in the prompt bar, you can double-click on the last vertex of the body outline to complete it.

Next, you include a reference designator (the rc07 text) on the Silkscreen layer.

14. Choose the **Setup > Text...** menu item, and fill in the dialog box as follows, then **OK** the dialog box. Leave all other fields at their default settings.

Height **0.06**
Orientation: **0**
Aspect Ratio: **1**
Stroke Width: **0.01**
Font: **std**
Justification: **Center Left**

15. **[Top Menu] Text > Add Reference Name:**

A prompt bar displays.

16. Check that the edit layer is still set to SILKSCREEN, then move the cursor into the Edit window. A box representing the reference text moves with the cursor. Position the text box so it is centered to the left of the resistor and click the Select mouse button.

The text for the reference designator now appears, as shown in Figure 3-15.

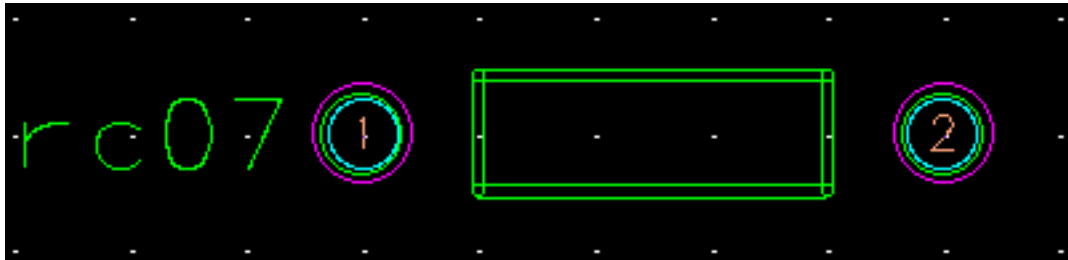


Figure 3-15. Location of the Reference Designator

For now, the displayed text equals the component geometry name. This name is known to the system because that is the name you specified when you created the geometry edit window. Later, the displayed text is replaced with a real reference designator, such as R57.

To complete this geometry you need to add a Placement Outline. The Placement Outline indicates to the Board Station tools the overall area a geometry requires during placement. The Placement Outline is automatically added to both place layers. In a later lab, when you place components on the board, the placement outline is used by LAYOUT to check for placement errors, such as component placement outlines overlapping. The size of the placement outline is very close the actual size of the component, or be slightly oversized. Refer to Figure 3-13 or Figure 3-16 for an illustration of the Placement Outline.

17. Choose the [Top Menu] **Attributes > Add Placement Outline > Both Layers:** menu item.

18. When the prompt bar is displayed, place the cursor where you will create the first corner of the placement outline, as shown in Figure 3-16, and click the Select mouse button. Click the Select mouse button once at the second corner of the placement outline, and once at the third corner. Finally, either double-click the Select mouse button at the final corner of the outline, *or* click once on the final corner of the outline and **OK** the prompt bar. **Cancel** the prompt bar if it repeats.

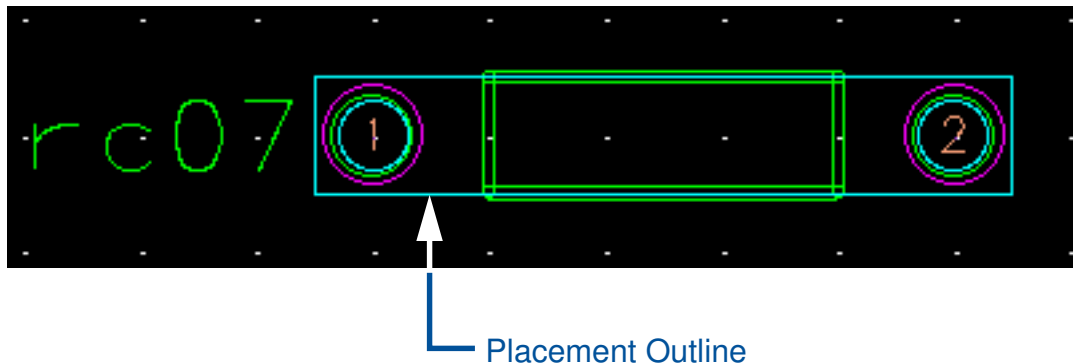


Figure 3-16. Resistor Placement Outline Added

To verify that the geometry meets the basic Board Station requirements, you will next run a check on the active parts.

19. Choose the **Check > Geometry > Active Geometry** menu item.

A message window reading *Geometry rc07 has no errors* appears on the screen

20. Close the message window.

If a warning or error appears, correct the problem before continuing.

You save the geometries later in the lab, after you have completed all of the components.

Creating the Capacitor Geometry

Now you create a geometry for capacitors. You follow almost the exact same procedure you used for the resistor. In this procedure, you are not provided with as detailed instructions as you were when you created the resistor geometry. If you need help, refer to the procedures you used when you created the resistor. Figure 3-17 shows the capacitor geometry you will create.

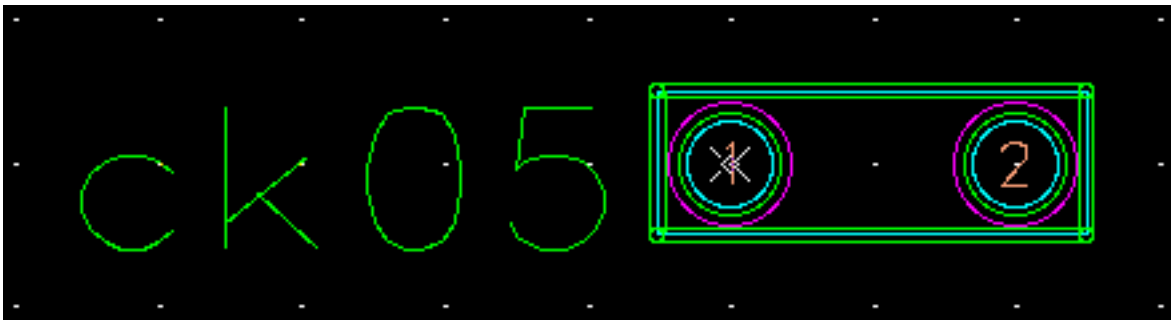


Figure 3-17. Completed Capacitor Geometry

1. Choose the **Geometries > Create Geometry > Component...** menu item.
2. Fill in the dialog box as follows, leaving all other information in the dialog box at their default values. When the dialog box is complete, **OK** it.

Component Name: **ck05**

Default Padstack Name: **th055028**

Another new edit window is created.

3. Activate the new edit window (Place cursor in edit window, and click Stroke mouse button).
4. Set up the grid for .05 inch spacing and a display interval of 2.
5. Verify the edit layer is SIGNAL_1.

6. Add the pin locations as you did for the resistor geometry. Be sure to specify the first pin is number 1. Place the first pin at absolute coordinate $x=0.0$, $y=0.0$. Place the second pin 0.2 inches to the right of the first pin. After you add both pins, Cancel the prompt bar.

Figure 3-18 shows how the pins are located relative to the grid and each other.



Figure 3-18. The Capacitor Pins

Next you add the capacitor's body outline.

7. Set up the edit layer to the SILKSCREEN layer.
If you need help setting the edit layer, refer to the procedure you used when creating the resistor.
8. Check the Line Width (also displayed in the upper-right corner of the window) to make sure it is set to 0.01. If it is not, set up the line width to 0.01.
9. Draw the capacitor body using the same method you used to create the resistor body outline. Refer to Figure 3-19 to see the shape and size of the capacitor body outline. When you finish drawing the body outline, cancel the Add Line prompt bar if it repeats.

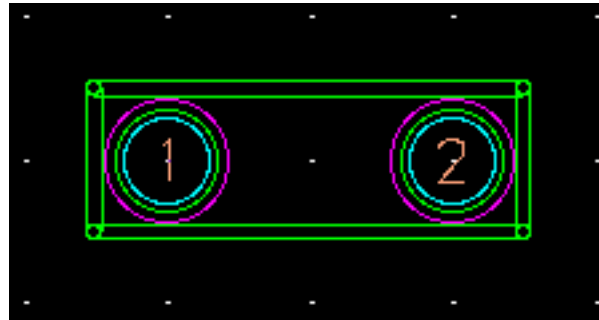


Figure 3-19. The Capacitor Body Added

10. Add the reference designator using the same procedure you used for the resistor's reference designator. Place the reference designator as shown in Figure 3-20.

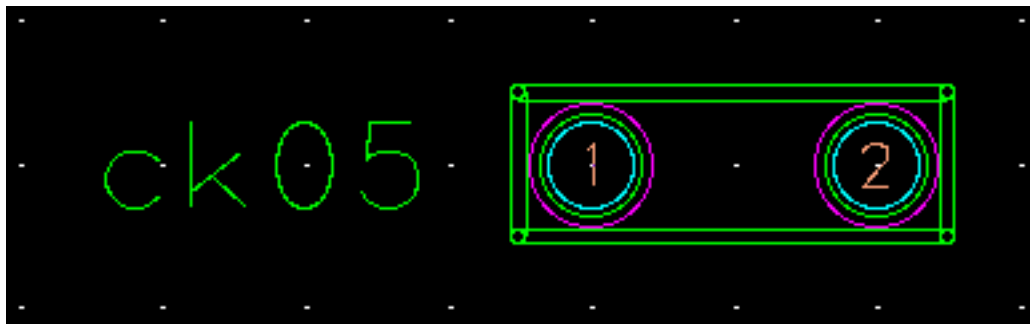


Figure 3-20. The Reference Designator Added

11. Complete the capacitor by adding the placement outline. Add the placement outline directly on top of the body outline, as shown in Figure 3-17, by clicking the Select mouse button on the four corners of the body outline.
12. Verify that the geometry meets the basic Board Station requirements by choosing the **Check > Geometry > Active Geometry** menu item. After you read the report and verify there are no errors, close the report window.

Again, if a warning or error appears, correct the problem before continuing.

Creating the Connector Geometry

You now know some of the basics for creating geometries. You are now ready for something a little more challenging. In this section, you build a 96-pin connector with pins on 0.1 inch spacing. You will use a pin array to automatically place all pins. You will have to define the component body and placement outline, add the reference designator, and add drill holes. You will also change the pin padstack for pin one of the connector to help identify it. The completed connector is shown in Figure 3-21.

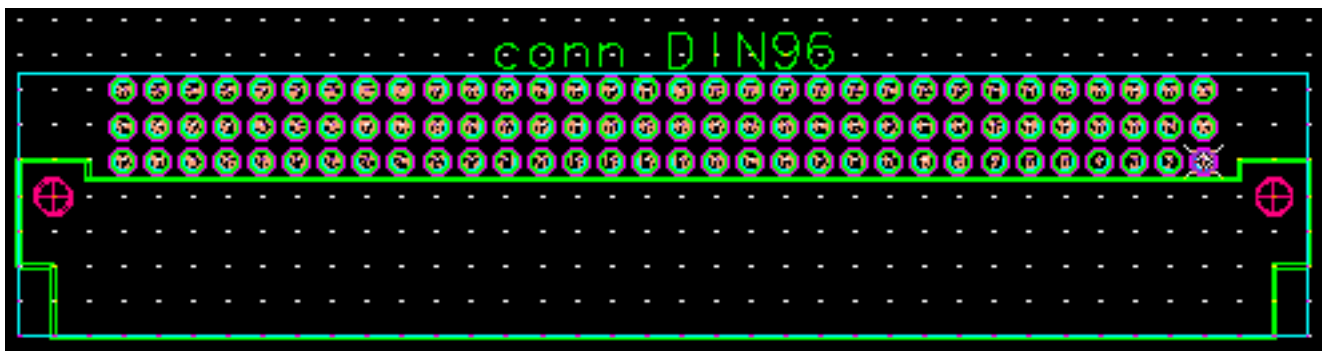


Figure 3-21. The 96 Pin Connector

1. Choose the **Geometries > Create Geometry > Component...** menu item. Complete the dialog box as follows, leaving all other options at their default values. When you finish, **OK** the dialog box.

Component Name: **conn_DIN96**

Default Padstack Name: **th055028**

Allow Component Outline to Overhang Board

2. Make sure the new edit window is active, then change the grid to 0.05 inch spacing with a display interval of 2.
3. Verify the edit layer is set to SIGNAL_1.

Next you add the pins. Because there are many pins, you do not want to add the pins one at a time, like you did with the capacitor and resistor geometries. Because the pins are evenly spaced, you can use a menu item that places the pins in an array, meaning the pins are placed automatically in rows (horizontally spaced), and columns (vertically spaced).

4. Choose the **[Top Menu] Pins > Add Pins Array...** menu item, and fill in the dialog box with the following. When you are done, **OK** the dialog box.

Alphas in Pin Name: **Not Included**

Starting Pin Number: 1

Pin Increment: 1

Padstack: [Leave this box blank]

Edit Page origin as starting location: **Yes**

Pin Orientation: **Row**

Number of Pins in a Row: **32**

Space between Pins: **0.1**

Sequence Pins from: **Right to Left**

Add Rows: **Bottom Up**

Number of Rows: 3

Space between Rows: 0.1

The pins are automatically placed starting at the origin and spaced in three rows 0.1 inches apart. The pins appear as shown in Figure 3-22.

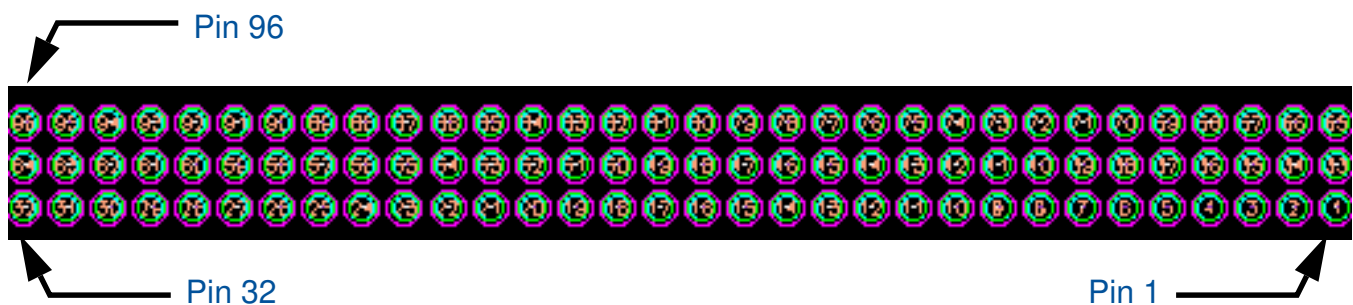


Figure 3-22. The Pins Array

The default option assigned a round pad (th055028, which is the default pin pad you defined for this component) to all pins. In this example, you change pin 1 to have a square pad, th055028sq.

5. View all the pins by using the View All stroke.

6. Choose the [**Top Menu**] **Attributes > Change Padstack Override > Change Padstack Override** menu item

A select area prompt bar displays.

7. Place the cursor on pin 1, and click the Select mouse button.

A prompt bar is displayed, and you are prompted for a Pad Override Name.

8. Enter **th055028sq** in the Pad Override Name field of the prompt bar, and **OK** the prompt bar.

Pin 1 now shows a square pad.

9. Unselect all items (press the Unselect All function key).

10. Set up the edit layer to be the SILKSCREEN layer and set up the line width to 0.01 inch.

11. Draw the connector body. Refer to Figure 3-23.

The outline of the connector appears as a double line because of the line width you set.

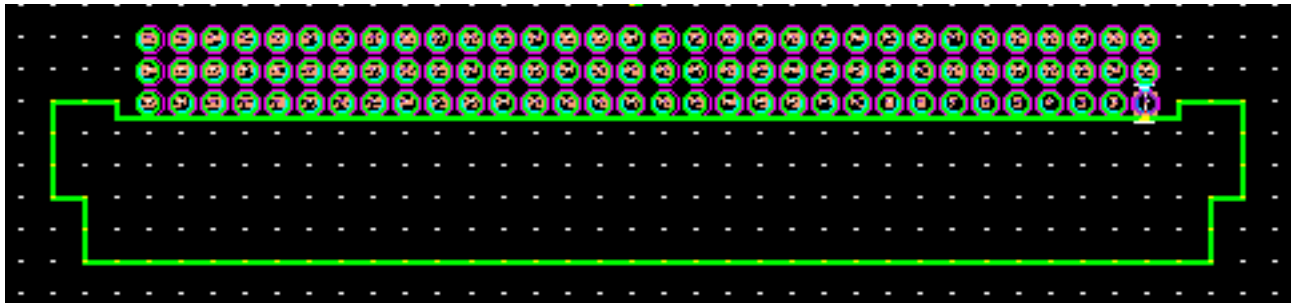


Figure 3-23. The Connector Outline Added

If you make a mistake, and you want to undo one or more line segments before you complete the connector outline, you can press the Backspace key to remove the last line segment. Each time you press the Backspace key, another line segment is removed.

Next you will add drill holes so the connector can be mounted to the board.

12. Choose the [**Top Menu**] **Shapes > Extended Menu > Add Drill Hole** menu item. In the dialog box, enter the **0.1** for the diameter. Choose **Options**. In the Options dialog box, choose Drill Hole Type: **unplated**, leave all other options default, and **OK** the dialog box. TAB to the location prompt. Place the cursor on the Absolute coordinate location X=-3.3, Y=-0.1, and click the Select mouse button.

The drill hole is added and is red because it is placed on the DRILL_HOLE layer. The prompt bar repeated so you can add another hole.

13. In the Add Drill Hole prompt bar that repeated, press the TAB key to highlight the location prompt, then place the cursor on Absolute coordinate location $X=0.2$, $Y=-0.1$, and click the Select mouse button. The two drill holes appear as shown in Figure 3-24.

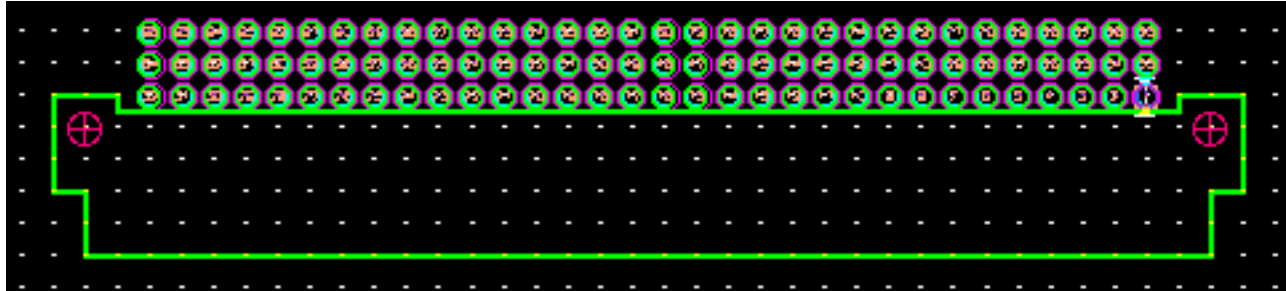


Figure 3-24. The Drill Holes Added

14. Complete the connector by adding the placement outline, as shown in Figure 3-25.

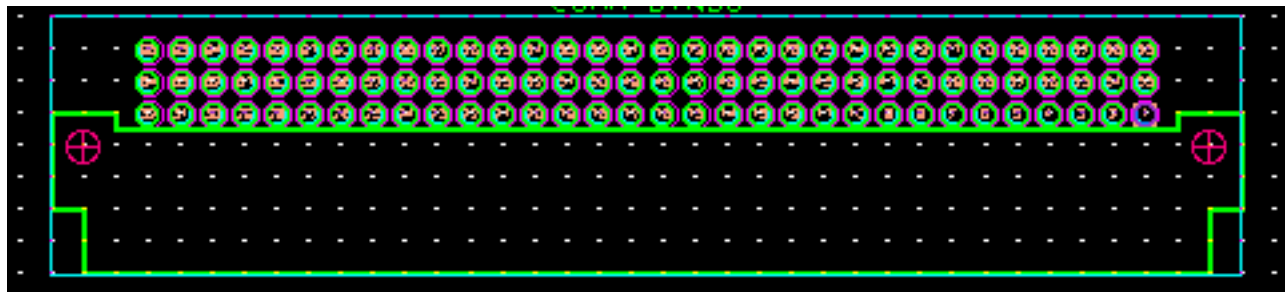


Figure 3-25. Connector with Placement Outline

Next you move the origin of the connector to the center of the right-side drill hole. Later, in the next lab, when you add the connector to the board, having the origin at the center of the hole helps you align the connector drill holes with the drill holes you put on the board.

15. Choose the **[Shapes] Move > Origin** menu item. When the prompt bar is displayed, choose the **[Shapes] Snap > Center** menu item. Place the cursor on the drill hole at the right side of the connector, and click the Select mouse button.

When you move the cursor, the origin is moved to the center of the drill hole, and the prompt bars are removed.

16. Add the reference designator above the connector, as shown in Figure 3-26.

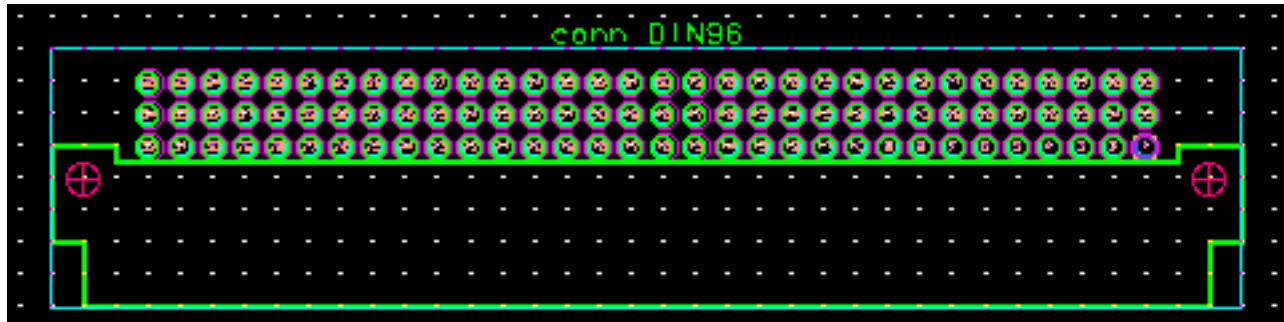


Figure 3-26. Complete Connector

17. Verify that the geometry meets the basic Board Station requirements by choosing the **Check > Geometry > Active Geometry** menu item.

Again, if a warning or error appears, correct the problem before continuing.

Creating a Surface Mount Component Geometry

There is one more geometry type to build and it is a surface mount part. The process for creating a surface mount geometry is nearly the same as it is for creating through-hole pin geometries. The differences occur in the choices you enter in the Create Component dialog box. Figure 3-27 shows the component you will build.

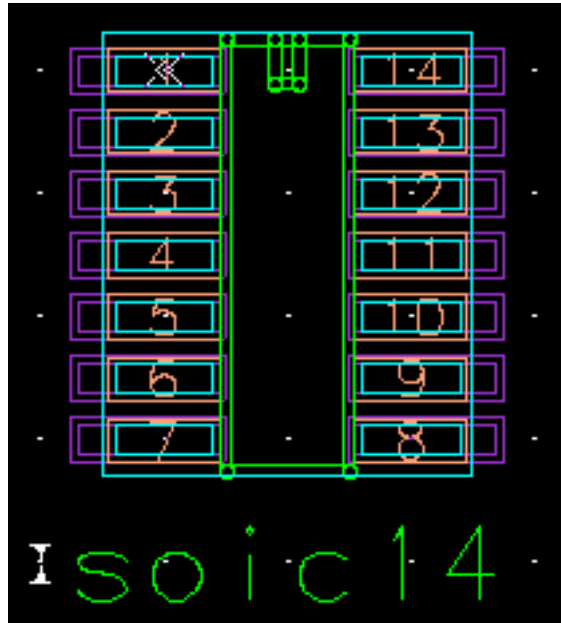


Figure 3-27. The Surface Mount Component

1. Choose the **Geometries > List Geometry Libraries** menu item. In the dialog box that displays, place the cursor on **USER =>** and click the Select mouse button.

A list of the user geometry libraries is displayed in the dialog box.

2. Place the cursor on **mgc.trng.padstacks**, and click the Select mouse button. After the dialog box indicates that the library is active, choose **View**.

The dialog box changes to show the contents of the mgc.trng.padstacks library.

3. From the list of padstacks, find s80X25 and s80X25right. Select both of them by placing the cursor on s80X25, holding down the Select mouse button, and dragging the cursor over s80X25right. Release the Select mouse button.

Both padstacks are highlighted.

4. Choose **Read** in the dialog box.

Two new edit windows are open, each contains one of the padstacks. These padstacks are now available for other components to use. When you create your component *soic14*, it uses these padstacks.

These pin padstacks each have a pad made for use on top of the board, and a pad for use on the back of the board. If the component that uses these pin padstacks is placed on the front/top of the board, the smaller pad_1 of each pin padstack is used. If the component that uses these pin padstacks is placed on the back of the board, the larger pad_2 of each pin padstack is used.

5. Choose the **Geometries > Create Geometry > Component...** menu item. Complete the dialog box as follows, leaving all other options at their default values. When you are finished, **OK** the dialog box.

Component Name: **soic14**

Default Padstack Name: *s80X25*

Both Surfaces

Surface Component

Turn off: **Allow Component Outline to Overhang Board**

The first difference is the type of default padstack. Here you have called for a padstack created specifically for surface mount devices. The graphics were created to appear on only a single side of the board.

The second difference is the definition of the geometry as a surface component. This definition sets an attribute that tells the Board Station tools how to place and connect to this component. You also set a switch specifying on which side or sides of the board this geometry can be placed.

6. Set up a grid spacing of .05 inches, with a display interval of 2.
7. Make sure the edit layer is SIGNAL_1.

Next you add the pins. Because of the order that the pins must be numbered, you are going to use a pin array (like you did for the connector), but you will use it once to create the pins on one side of the component, and then you will use it again to create the pins on the other side. First you will add pins 1 through 7.

8. Choose the [Top Menu] Pins > Add Pins Array... menu item, and fill in the dialog box with the following. When you are done, **OK** the dialog box.

Alphas in Pin Name: **Not Included**

Starting Pin Number: **1**

Pin Increment: **1**

Padstack: **s80X25**

Edit Page origin as starting location: **Yes**

Pin Orientation: **Column**

Number of Pins in a Column: **7**

Space between Pins: **0.05**

Sequence Pins from: **Top Down**
One Column

Now you add pins 8 through 14. This time, you specify a different pin padstack, which is also a surface pad, and you specify the location of pin 8.

9. Choose the [Top Menu] **Pins > Add Pins Array...** menu item, and fill in the dialog box with the following. When you are done, **OK** the dialog box.

Alphas in Pin Name: **Not Included**

Starting Pin Number: **8**

Pin Increment: **1**

Padstack: **s80X25right**

Edit Page origin as starting location: **No**

Pin Orientation: **Column**

Number of Pins in a Column: **7**

Space between Pins: **0.05**

Sequence Pins from: **Bottom Up**

One Column

You are prompted for a location, which you supply in the next step.

10. Place the cursor at Absolute coordinate X=0.2, Y=-0.3, two visible grid points to the right of pin 7, and click the Select mouse button.

The pins appear as shown in Figure 3-28.

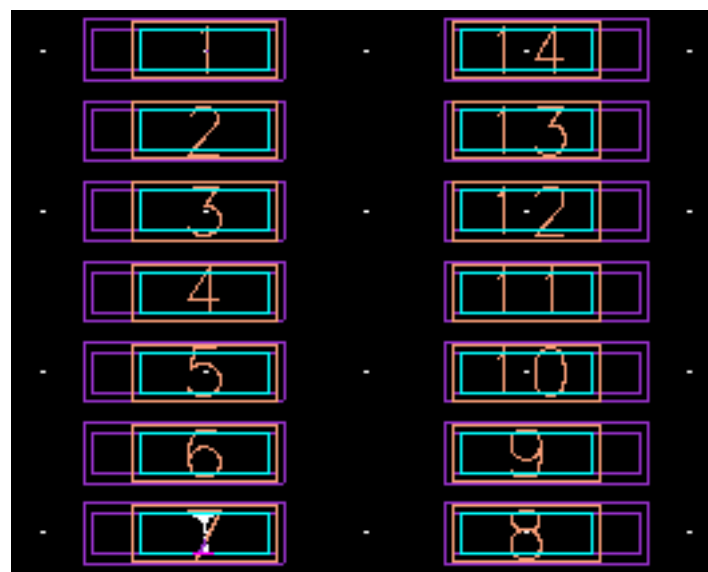


Figure 3-28. The Pins of SOIC14

11. Change the grid spacing to 0.025 inch, with an interval of 1 to make it easier to add the component outline graphics.
12. Change the edit layer to Silkscreen, then add the component outline as shown in Figure 3-29.

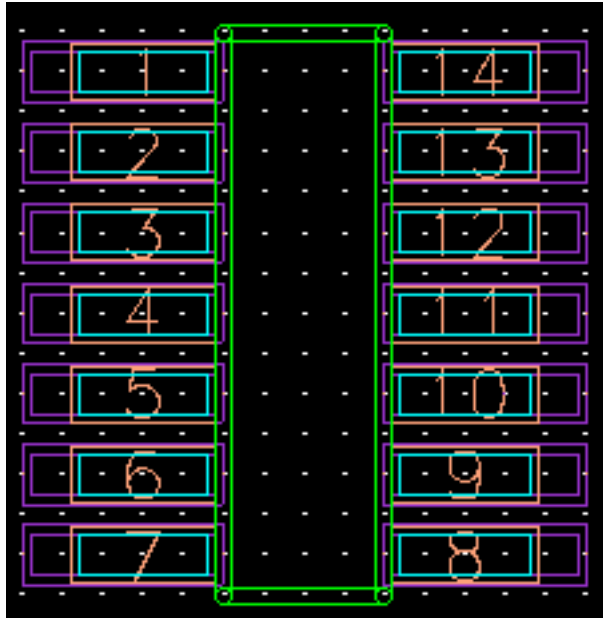


Figure 3-29. The Component Outline Graphics Added to SOIC14

13. Add the reference designator and placement outline as shown in Figure 3-30.

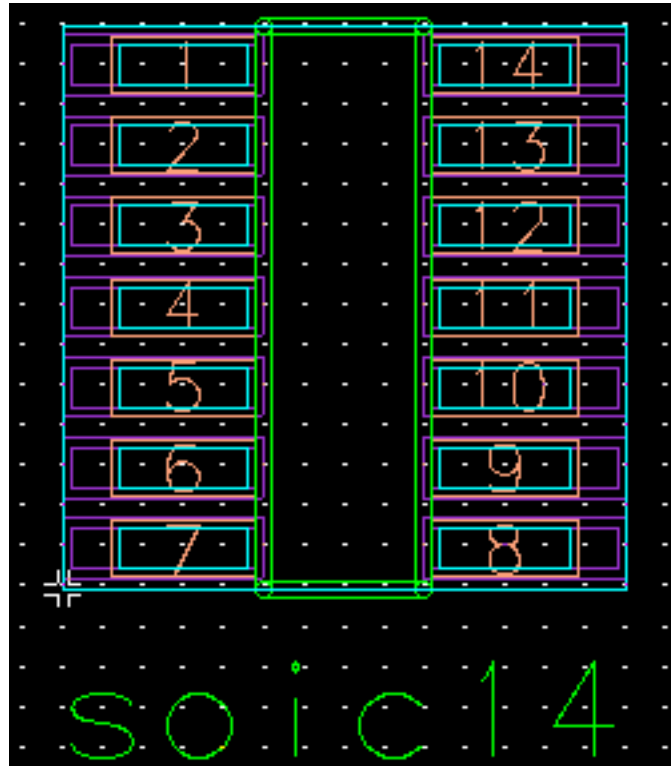


Figure 3-30. The Placement Outline and Reference Designator on SOIC14

You add the notch in the next step. Do not do it now, because you first need a finer grid spacing.

14. Set the grid spacing to 0.0125 inch, with a display interval of 4, then add the notch to the top of the body outline, as shown in Figure 3-31.

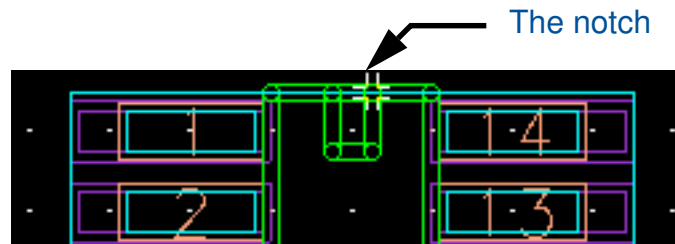


Figure 3-31. The Notch added to the SOIC14 Component

You can build surface mount components such as the SOIC with the origin on pin 1 or in the center for auto-insertion equipment. You might find it easier to build your components with the origin at pin 1 and then use the function: [Top Menu] Shapes > Move > Origin: to move the origin.

15. Verify that the geometry meets the basic Board Station requirements by choosing the **Check > Geometry > Active Geometry** menu item.

Again, if a warning or error appears, correct the problem before continuing.

Saving Geometries and Leaving LIBRARIAN

Now that these geometries have been created, you need to save the geometries somewhere so that they can be used later. You save them in the **trng** library you created when you saved the logo and card_eject geometries during a previous lab.

Saving Geometries in a Workshop

If you are completing this training in an instructor-led workshop, use this procedure section to save your geometries. If you are doing this training in a self-paced Personal Learning Program, skip this procedure and complete section "Saving Geometries in a Personal Learning Program" on page 3-50.

1. Choose the **File > Save > Save ASCII Geometries...** menu item. Fill in the dialog box as follows, then **OK** the dialog box:

Geometries to Save: **All Geometries**
Separate File For Each
Library to Store the Geometry: **Other**
Directory Pathname: your_path/**pcb_parts/user_geom/trng**
Replace Existing File(s)

You chose to replace existing files even though you are creating a new directory (trng). You did this just to make certain the new directory is created.

The *Other* library is any place you want to store personal or temporary geometry when you do not want to place it in an official library.

A Report-Message window appears confirming that the files were written and that they were written to the correct location.

2. Close the report-message window.

Saving Geometries in a Personal Learning Program

Complete this procedure section only if you are doing this training as part of a self-paced Personal Learning Program.

1. Choose the **File > Save > Save ASCII Geometries...** menu item. Fill in the dialog box as follows, then **OK** the dialog box:

Geometries to Save: **All Geometries**

Separate File For Each

Library to Store the Geometry: **Other**

Directory Pathname:

your_path/training/board_new/mod3/sig_az/pcb_parts/
user_geom/trng

Replace Existing File(s)

There is a pcb_parts directory that is provided for you in your training design data directory. You wrote the trng file in that directory instead of into your \$HOME/pcb_parts directory so that the training data is not mixed up with your permanent pcb_parts directory. The system records the name, creates the directory, and saves the geometries at the directory pathname you specified.

The *Other* library is anyplace you want to store personal or temporary geometry when you do not want to place it in an official library.

You chose to replace existing files even though you are creating a new directory (trng). You did this just to make certain the new directory is created.

A Report-Message window appears confirming that the files were written and that they were written to the correct location.

2. Close the report-message window.

Closing LIBRARIAN

Close the LIBRARIAN session by choosing **Close** from the Window Menu (upper-left corner icon). When the *Save change to design?* dialog box appears, choose **No**.

You choose no because:

- You have already saved the geometries to a **trng** directory where they can be accessed for any new design.
- You did not enter LIBRARIAN with a design so there is no active design in which to save them.

Congratulations! You have completed Lab 3: "Creating Component Geometries". The next lesson is Lesson 4: "Creating Board Geometries".

Lesson 4

Creating Board Geometries

Now you study how to create a geometry for the board and save it with a design.

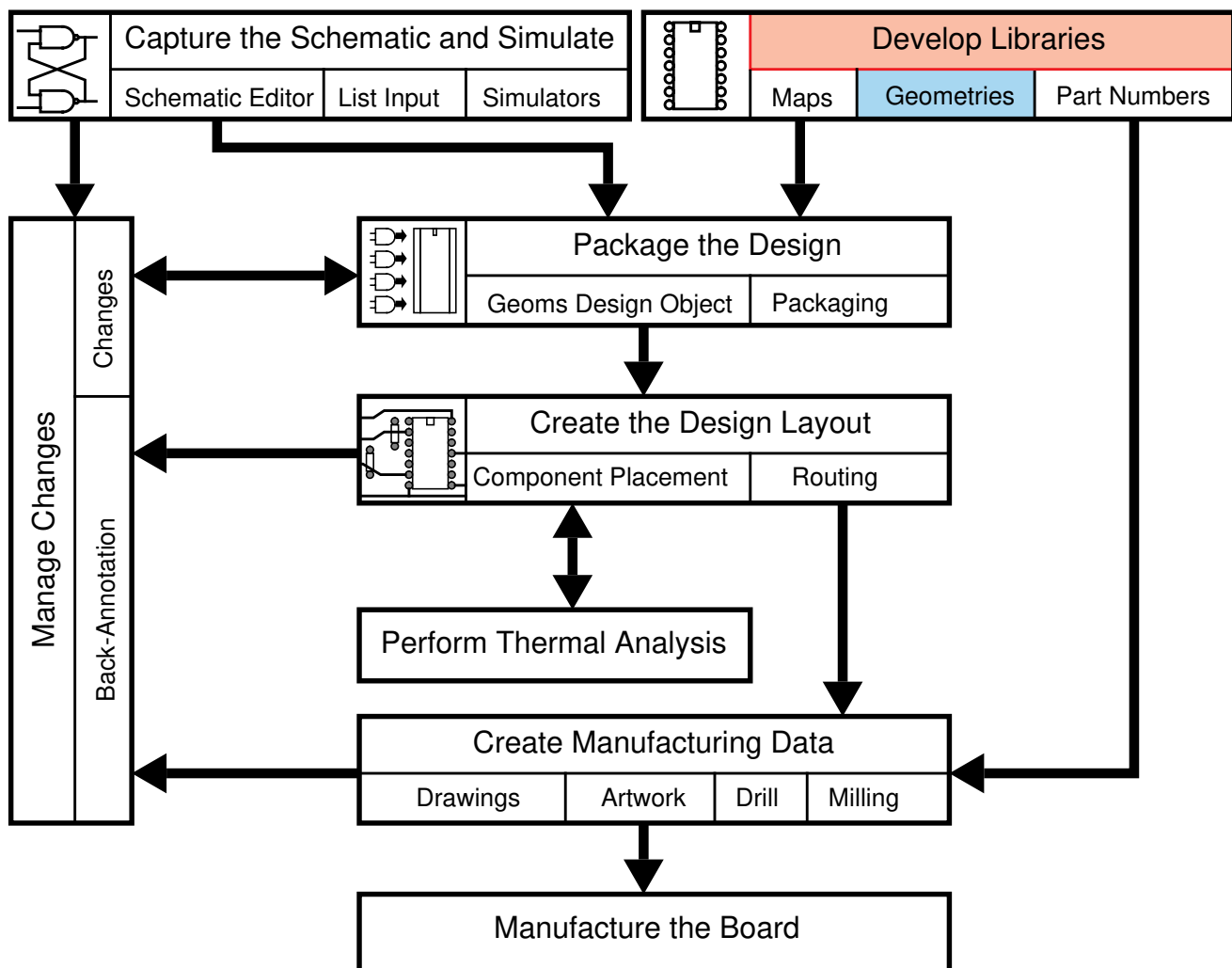


Figure 4-1. PCB Design Process

Objectives

In this module you learn about the board geometry. The board geometry is the most important element of your design because it defines the framework for placing components and routing traces, and it defines some of the manufacturing requirements for your design. At the end of this lesson, you can:

- Start a design by creating a *pcb* container.
- Describe the process for creating the board geometry.
- Describe the function of the *tech* design object.

Process for Creating a Board

To create a board, use the following process. Each step of the process is discussed in detail in later sections of this lesson. In the lab exercise, you will follow this process.

1. Create a design. A board is associated with a design, so you need to create a design container, which is a directory, to hold your design data. For more information, refer to section “Starting a Design” on page 4-3.
2. Invoke LIBRARIAN on the design. From the shell, use the following syntax:
librarian design_name
You can also invoke LIBRARIAN on a design from the Design Manager.
3. Choose the **Geometries > Create Geometry > Board...** pulldown menu item in LIBRARIAN, and enter the board attributes in the dialog box. For more information, refer to section “Board Attributes” on page 4-4.
4. Create the board geometry, including its outline, mounting holes, attached hardware, logos, and so on. Use the geometry creation techniques you learned in previous lessons and labs of this module.
5. Add other attributes (using the [**Top Menu**] **Attributes** popup submenu) to define the placement outline, the routing outline, and so on. For more information, refer to section “More Board Attributes” on page 4-8.

6. Specify the design rules, including the physical layers, net rules, and so on. For more information, refer to section “Design Rules” on page 4-10, and section “Physical Layers” on page 4-12.
7. Save the board with the design and, if you want to, also save the board as an ASCII geometry as a form of backup.

Starting a Design

To start a design you need a design container (directory) that includes a *pcb* container. If you are using a Design Architect schematic, the design container already exists.

The following description explains the process for creating a design container and a *pcb* container, if you are not using a Design Architect schematic.

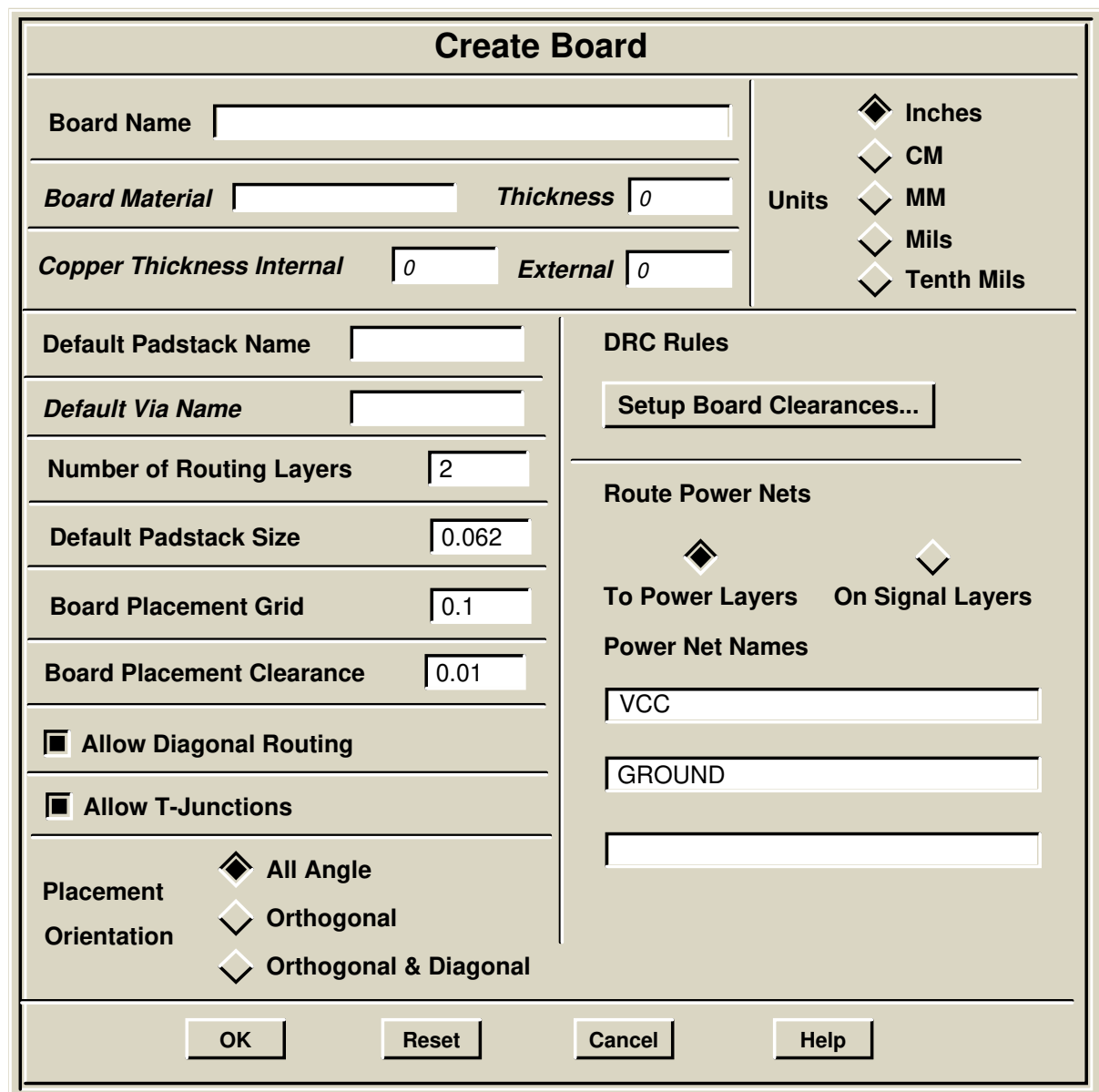
1. Create a directory in which to place all of your design data. This directory will later become a PCB design object. To create a directory from the Design Manager, open a navigator window. Then, choose **Add > Directory:** from the menu bar.
Enter the pathname of the directory in the entry box of the prompt bar. Press the OK button in the prompt bar to create the directory.
2. Invoke the LIBRARIAN tool on the design. When you double-click on the LIBRARIAN icon in the Tools window in the Design Manager, a dialog box is displayed. You choose *On a Design* in the dialog box to open LIBRARIAN on a design. When you *OK* the dialog box, a navigator dialog box is displayed in which you navigate to the design you want to open.

LIBRARIAN examines the contents of the design directory. If an attribute file, which makes the directory a PCB design object, is missing, one is created automatically. LIBRARIAN also creates a *pcb* container and a *mfg* container inside the PCB design object, if needed. The *design_geom*, *design_maps*, and *design_lib* directories are also automatically created, if needed.

Remember, the operating system commands and file system object names are case sensitive. The PCB tools generally are not case sensitive except when the tools deal with the operating system, such as referring to filenames or pathnames. The effects of case sensitivity are important when accessing any data stored in a file from any Board Station tool.

Board Attributes

Board attributes are added to the geometry automatically when you choose the **Geometries > Create Geometry > Board...** menu item and fill in the Create Board dialog box, as shown in Figure 4-2.



The "Create Board" dialog box is a graphical user interface for configuring board attributes. It features a title bar "Create Board" and a main area with various input fields and options. The layout is organized into sections: Board Name, Board Material, Thickness, Copper Thickness Internal/External, Units, Default Padstack Name, Default Via Name, Number of Routing Layers, Default Padstack Size, Board Placement Grid, Board Placement Clearance, Allow Diagonal Routing, Allow T-Junctions, Placement Orientation, DRC Rules, Route Power Nets, and Power Net Names. At the bottom are buttons for OK, Reset, Cancel, and Help.

Create Board	
Board Name <input type="text"/>	Units <input checked="" type="radio"/> Inches <input type="radio"/> CM <input type="radio"/> MM <input type="radio"/> Mils <input type="radio"/> Tenth Mils
Board Material <input type="text"/> Thickness <input type="text" value="0"/>	
Copper Thickness Internal <input type="text" value="0"/> External <input type="text" value="0"/>	
Default Padstack Name <input type="text"/>	DRC Rules <input type="button" value="Setup Board Clearances..."/> Route Power Nets <input checked="" type="radio"/> To Power Layers <input type="radio"/> On Signal Layers Power Net Names <input type="text" value="VCC"/> <input type="text" value="GROUND"/> <input type="text"/>
Default Via Name <input type="text"/>	
Number of Routing Layers <input type="text" value="2"/>	
Default Padstack Size <input type="text" value="0.062"/>	
Board Placement Grid <input type="text" value="0.1"/>	
Board Placement Clearance <input type="text" value="0.01"/>	
<input checked="" type="checkbox"/> Allow Diagonal Routing	
<input checked="" type="checkbox"/> Allow T-Junctions	
Placement <input checked="" type="radio"/> All Angle	
Orientation <input type="radio"/> Orthogonal	
<input type="radio"/> Orthogonal & Diagonal	
<input type="button" value="OK"/> <input type="button" value="Reset"/> <input type="button" value="Cancel"/> <input type="button" value="Help"/>	

Figure 4-2. Create Board Dialog Box

The options you can specify include:

- **Board Material**—adds the attribute `Board_material`, which identifies the board construction material for use in thermal calculation. `Board_material` is a thermal attribute used in `AutoTherm`.
- **Thickness**—adds the attribute `Board_thickness`, which identifies the board thickness for use in thermal calculations.
- **Copper Thickness Internal**—adds the attribute `Board_internal_copper`, which identifies the copper thickness of internal layers for use in thermal calculations.
- **External**—adds the attribute `Board_external_copper`, which identifies the copper thickness of external layers for use in thermal calculations.
- **Default Padstack Name**—adds the attribute `Board_default_padstack`, which defines the padstack used by all the pins on the board unless a component specifies a padstack with the `Component_default_padstack` or `Component_padstack_override` attribute for the component's pins.
- **Number of Routing Layers**—adds the attribute `Board_routing_layers`, which defines the default physical layer structure used if you enter `LAYOUT` without technology data in the *tech* design object. This number does not include power layers.
- **Default Padstack Size**—adds the attribute `Default_pad_size`, which specifies the default pad diameter. When you specify a non-uniform routing grid, the autorouter in `LAYOUT` uses the `Default_pad_size`, along with the default net type rules to determine the grid spacing.
- **Board Placement Grid**—adds the attribute `Board_placement_grid`, which specifies the placement grid used for placing components on the board in `LAYOUT`. All component origins are located on this grid.

- **Allow Diagonal Routing**—adds the attribute `Diagonal_routing_allowed`, which enables or disables diagonal routing. By default, the `Diagonal_routing_allowed` attribute is set to *yes*. Unselect **Allow Diagonal Routing** to disable diagonal routing on the board.
- **Allow T-Junctions**—adds the attribute `Tjunctions_allowed`, which enables or disables t-junctions during routing. By default, the `Tjunctions_allowed` attribute is set to *yes*. Unselect **Allow T-Junctions** to prohibit t-junctions when routing a board in LAYOUT.
- **Placement Orientation All Angle/Orthogonal/Orthogonal & Diagonal**—adds the attribute `Orthogonal_placement_only` or `Diagonal_placement_allowed`. If you do not specify the `Orthogonal_placement_only` attribute or the `Diagonal_placement_allowed` attribute, by default, all-angle placement is permitted on the board. The `Component_diagonal_allowed` and `Component_orthogonal_only` attribute take precedence over the board attributes.
- **DRC Rules**—pressing the Setup Board Clearances button provides another dialog box in which you can specify the following board clearances:
 - **Drill Hole Clearance**—determines the minimum distance between a drill hole and the board edge.
 - **Routing Outline Clearance**—determines the minimum distance between the routing outline and the board edge.
 - **Placement Outline Clearance**—determines the minimum distance between the placement outline and the board edge.
 - **Copper Clearances by Layer**—determines the minimum distance between the board edge and paths, polygons, arcs, circles, the bounding box of text, and the bounding box of an added part. You can specify separate copper clearance values for individual logical layers.

The default clearance for all cases is .001. If you specify a different clearance for any of the objects, the new clearance is stored in the *tech* design object. After a clearance is established, you receive a warning message if any clearances are violated. For example, if you specify that the routing outline clearance is 0.025, and you create a routing outline that is offset from the board outline by only 0.01, the system issues a warning message.

Objects placed outside the board outline are not checked for clearance violations.

- **Route Power Nets To Power Layers/On Signal Layers**—adds the attribute `Power_net_names`, which specifies the names of the power nets associated with the internal planes for a multi-layer design.

You enter the names of the power and ground nets that are in your schematic that you want on the power layers. Use the same case for the names as they appear in your schematic (if the names are uppercase, use uppercase).



Split Power Plane

If you are creating a board with split power planes (two or more power or ground nets on a single physical layer), enter the power and ground nets in sequence. For example, if you want a power layer with power nets VCC and POS15V, and another power layer with ground nets GROUND and NEG15V, you enter the net names that will be on the same layer in sequence in the list, such as: VCC, POS15V, GROUND, NEG15V. VCC will be automatically assigned to layer power_1, POS15V will be assigned to power_2, GROUND will be assigned to power_3 and NEG15V will be assigned to power_4. Later, when you create the Artwork Order, you assign the physical power layers that go together on a split power plane to the same Artwork Layer. In this example, you would assign power_1 (VCC) and power_2 (POS15V) to a single artwork layer. You would assign power_3 and power_4 to another artwork layer. You will create a board with a similar split power plane arrangement in the lab exercise following this lesson.

If you choose **On Signal Layers**, the attribute is added with the net name argument of `no_power_layers`.

More Board Attributes

Additional board attributes are added from the **Attributes** popup menu, as shown in Figure 4-3. The **Attributes** popup menu contains these items only when a board geometry edit window is active.

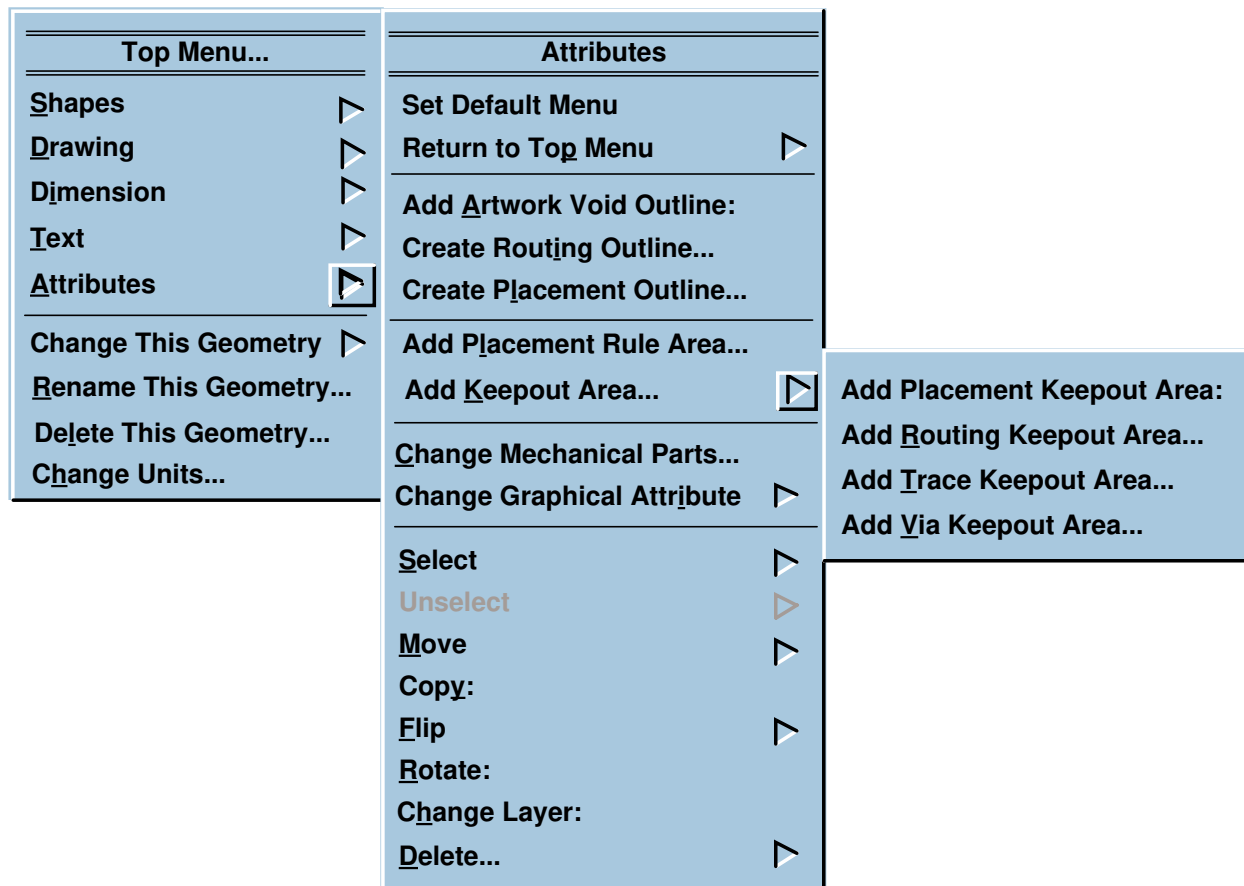


Figure 4-3. Attributes Popup Menu

- **Add Artwork Void Outline**—adds the attribute `Artwork_void`, which defines the shape of a void area around the board image or other features, such as registration marks. You use this attribute if the manufacturing process for your board requires the panel to include thieving patterns (crop marks).
- **Create Routing Outline**—adds the attribute `Board_routing_outline`, which defines a polygon where routing is allowed on the board.

- **Create Placement Outline**—adds the attribute `Board_placement_outline`, which defines the area where components may be placed on the board in LAYOUT.
- **Add Placement Rule Area**—adds the attribute `Board_placement_region`, which designates an area as the placement region for a particular circuit group from the schematic or applies a component height restriction to the area.

You can use this attribute, for example, to create an area that will contain only analog parts. Another use is to create an area that contains only parts that use a specific power or a specific ground, such as when you create a board with split power planes.

- **Add Placement Keepout Area**—adds the attribute `Board_placement_keepout`, which specifies a keepout area within the board definition for component geometries. A specific side of the board can be specified as the placement layer.
- **Add Routing Keepout Area**—adds the attribute `Routing_keepout`, which describes an area on the geometry where routing is restricted.
- **Add Trace Keepout Area**—adds the attribute `Trace_keepout`, which describes an area on the geometry where traces are not allowed. Vias can be added within a trace keepout area.
- **Add Via Keepout Area**—adds the attribute `Via_keepout`, which describes an area on the geometry where vias are not allowed.
- **Add Dimension Keepout Area**—adds a rectangular area to a geometry, either surrounding the geometry, or to one side of the geometry. The area prohibits the placement of horizontal and vertical dimensions within the area if the `clearance_check` optional switch of the `$add_horizontal_dimension()` or `$add_vertical_dimension()` functions is set when you add the dimension.
- **Change Mechanical Parts**—adds the attribute `Mechanical_parts`, which associates non-electrical part information with the board geometry. Mechanical parts include heat sinks, screws, lock washers, and card ejectors.

Other Board Features

You can add many additional features to board geometry. Many of these features are used or included in the manufacturing data that is produced. Some users prefer to add the manufacturing related features to their board geometry when they prepare their manufacturing data. The choice depends on your design processes. Consider that a board geometry containing standard manufacturing features, like targets and dimensions, requires less time to prepare for manufacturing data generation when it is used in several designs.

Features like targets, logos, and crop marks are created as generic geometries using the techniques that you learned earlier in the course. You can then add these generic geometries to the board geometry. Dimensions can be added in LIBRARIAN or FabLink. The techniques for adding dimensions are covered in section "Adding Dimensions".

Design Rules

Design rules are values that control several important aspects of both interactive and automatic routing in LAYOUT, and design rules form the basis for checking the connections created by the routers in LAYOUT. You can set up Design rules either in LIBRARIAN or in LAYOUT. Design rules include attributes assigned to the board geometry, such as Board_routing_outline and Routing_keepout, which define the boundaries within which routing can occur.

The categories of design rules are:

- **Physical layer rules**—define the stacking order of the physical board layers.
- **Pin rules**—identify the layers spanned by the padstack, designate layers on which to prevent connections, and specify whether the pin rule applies to top or bottom placement of a component.
- **Via rules**—identify the layers spanned by the padstack and designate the layers on which the via padstack can connect directly to a single-layer pin padstack.

- **Layer rules**—allow or suppress routing on a layer and set the preferred routing direction on a layer.
- **Net rules**—are design rules like trace width, pin, via, and trace clearances that apply to the nets in a design. Assigned by user-defined net types, using the Net_type property.
- **Net rules for layers**—are design rules that apply to nets on a specific layer.
- **Drill hole clearance**—determines minimum distance between a drill hole and the board edge. Included in check geometries, artwork creation checks, and a specific padstack clearance check.
- **Routing outline clearance**—determines minimum distance between the routing outline and the board edge.
- **Placement outline clearance**—determines minimum distance between the placement outline and the board edge.
- **Copper clearances**—determines minimum distance between the board edge and paths, polygons, arcs, circles, the bounding boxes of text and added parts. You can specify the copper clearance for individual logical layers.

The *tech* design object stores the design rules and clearances. To review all of the existing design rules for your design, choose the **View > Design Data** menu item from the menu bar and, from the resulting list of design objects, select the *tech* design object and press the OK button. For a detailed discussion of design rules, refer to section "Design Rules" in Module 7: *Routing Traces on a Circuit Board*.

Physical Layers

By default, the system creates a physical layer structure based on the signal and power layers defined for your board geometry. This structure is automatically saved in the *tech* design object during any normal exit from LAYOUT, and restored whenever LAYOUT is invoked.

To modify the default physical layer structure, choose the **Setup Design Rules > Physical Layers...** menu item. A dialog box is displayed, as shown in Figure 4-4.

The Order No. column is the order in which LAYOUT toggles through layers when routing traces.

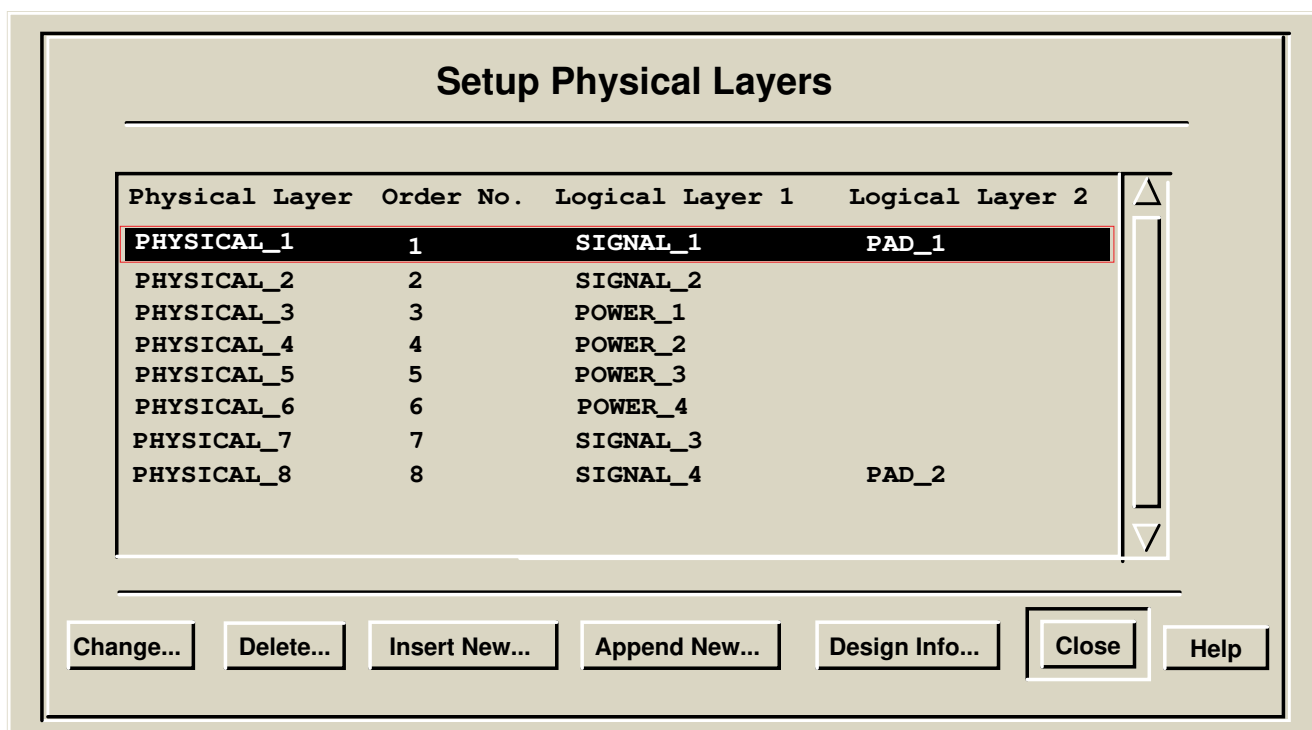


Figure 4-4. Setup Physical Layers Dialog Box

Default Physical Layers

Physical layers are numbered from 1, for the top surface layer, to n , for the bottom surface layer. The n represents the stacking order number of the bottom layer. The default physical layer names use the stacking order number, such as Physical_1 for the top physical layer, Physical_2 for the second physical layer in the stacking order, and ending with Physical_ n for the bottom physical layer. With the exception of the top and bottom (surface) physical layers, only one logical layer is associated with each physical layer. The default scheme assigns the logical signal layer and power layer names to physical layers; for example, Signal_2 to Physical_2. For the top and bottom layers, the system also assigns the Pad_1 and Pad_2 logical layers. Refer to Figure 4-5.



When assigning power layers to physical layers, be sure to assign the power layers sequentially.

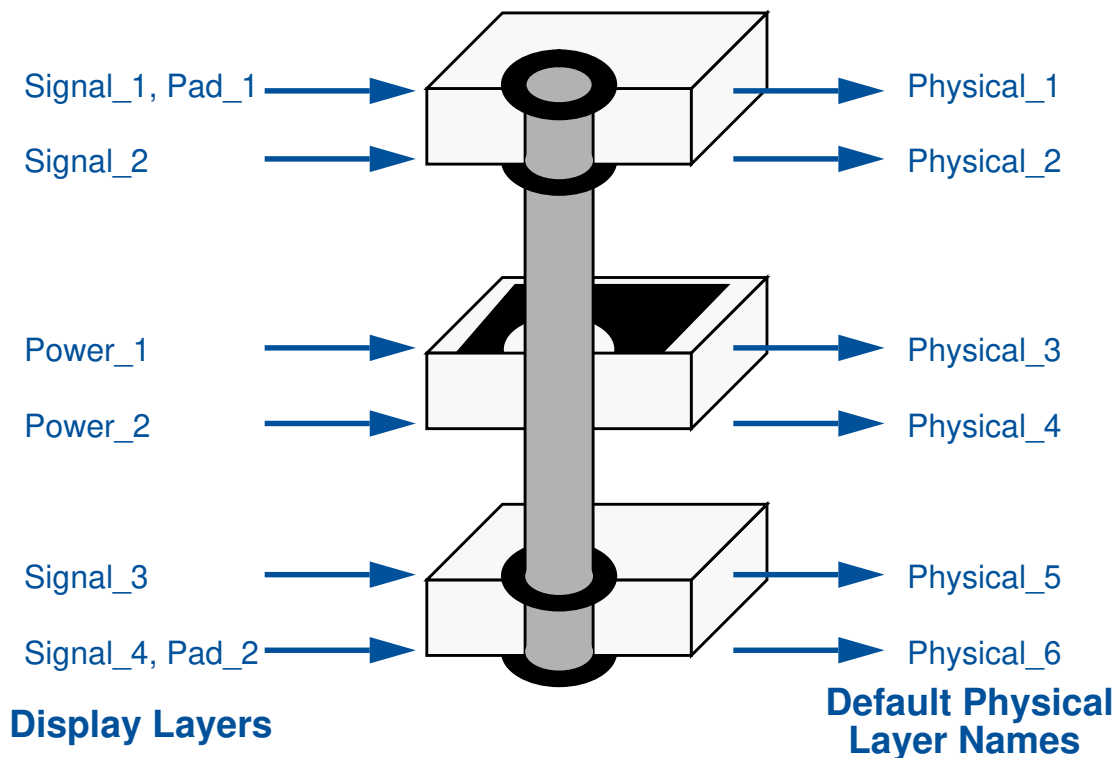


Figure 4-5. Default Physical Layers

Lab Exercise

In this lab exercise you create a multi-layer double-sided board with split power and ground planes. You add placement and routing outlines, mounting holes, and extractor tabs. Upon completion of this lab exercise you can:

- Create a design directory and invoke LIBRARIAN on a design.
- Specify a board's attributes, create its outline geometry, and add a placement outline, a routing outline, and a placement region for special components.
- Add mounting holes.
- Add non-electrical geometries such as a logo and card ejectors.
- Set up Design Rules.
- Save the board with the design.

Turn to Module 3—Lab 4: "Creating Board Geometries".

Lab 4

Creating Board Geometries

Introduction

In this lab exercise you explore the techniques for creating a board geometry. You explore the construction of the outline, the definition of the routing and placement outlines, and the addition of other features.

Upon completion of this lab exercise, you can:

- Create a design directory and invoke LIBRARIAN on a design.
- Specify a board's attributes, create its outline geometry, and add a placement outline, a routing outline, and a placement region for special components.
- Add mounting holes.
- Add electrical geometries such as a connector.
- Set up Design Rules.
- Save the board with the design.

Procedure

In this procedure, you invoke LIBRARIAN on a specific design to create board geometries.

Preparation for Lab

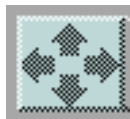
In the previous labs you have been invoking LIBRARIAN independent of a design. In this lab, you create a board for a specific design. This lab assumes that a design directory is created for you. If you receive Design Architect schematic data to make your board design, the design directory is created for you (by Design Architect). If your schematic data does not come from Design Architect, you will not have a design directory, and you will have to make one (using either the Add > Directory menu item in the Design Manager, or the mkdir command in a UNIX shell). The labs in this training come with a design directory created by Design Architect.

Invoke the Design Manager and LIBRARIAN.

1. Invoke the Design Manager.

Refer to the previous lab if you forgot how to invoke the Design Manager.

2. Using the Design Manager, change your current directory to the *mod3* directory (where the design is located), by clicking on the Change Directory icon in the navigator window, as shown in Figure 4-6. In the *Change directory to* dialog box that displays, enter the pathname: `your_path/training/board_new/mod3` and press the RETURN key.



Click the Select mouse button on this icon to change directories

Figure 4-6. Change Directory Icon

3. Invoke LIBRARIAN by placing the cursor on the LIBRARIAN icon and double clicking the Select mouse button.
4. In the Specify Invocation Mode dialog box that appears, choose **Invocation Mode: On a Design**. Then press the **OK** button in the dialog box.

A navigator dialog box is displayed.

5. In the navigator dialog box, click on **sig_az** to choose it, then **OK** the dialog box.

Another dialog box is displayed, and you are prompted for the technology to use for this design.

6. In the dialog box, choose **Standard PCB** for the technology, and **OK** the dialog box.

Another shell is created, which will contain the transcript for LIBRARIAN. Soon, the LIBRARIAN session window is displayed. A Report-Startup message might appear in the middle of the LIBRARIAN Session window.

7. After reading the report notes, close the report window, and then maximize the size of the LIBRARIAN session window to fill the display.

The design directory *sig_az*, is a directory created by Design Architect to contain a schematic. You will add many pcb design objects to the *sig_az* directory. In the prior labs, you created parts and saved them in ASCII format in a User parts library. These parts library parts can be used in any design. Now you will create a board, and add some parts from the parts libraries to the board. When you finish creating the board, you will save two copies of the board and its parts. One copy will be in ASCII format in the parts libraries, so it can be used by other designs if you wish, and you will save another copy inside the *sig_az* design directory in binary format specific to the *sig_az* design. You will find that as the labs progress, you will be reading parts from the parts libraries, and saving them to the *sig_az* design. This is how you take general purpose library parts and make them specific to a design; by reading the parts from a library, and saving them to a design.

You can only save parts to a design if you first open the design with a Board Station application, such as Librarian. If you do not open an application on a design, you cannot save to a design.

Creating the Board Geometry

In this procedure, you create a board geometry. The board geometry represents the overall size and shape of the board. It contains the routing and placement characteristics and it might also contain additional hardware information. In the next few steps, you create the board geometry edit window and define attributes for the board. After that, you use the geometry creation techniques you learned in the previous labs to create the board outline shown in Figure 4-7. Use this figure and its dimensions as a reference when you create your board.

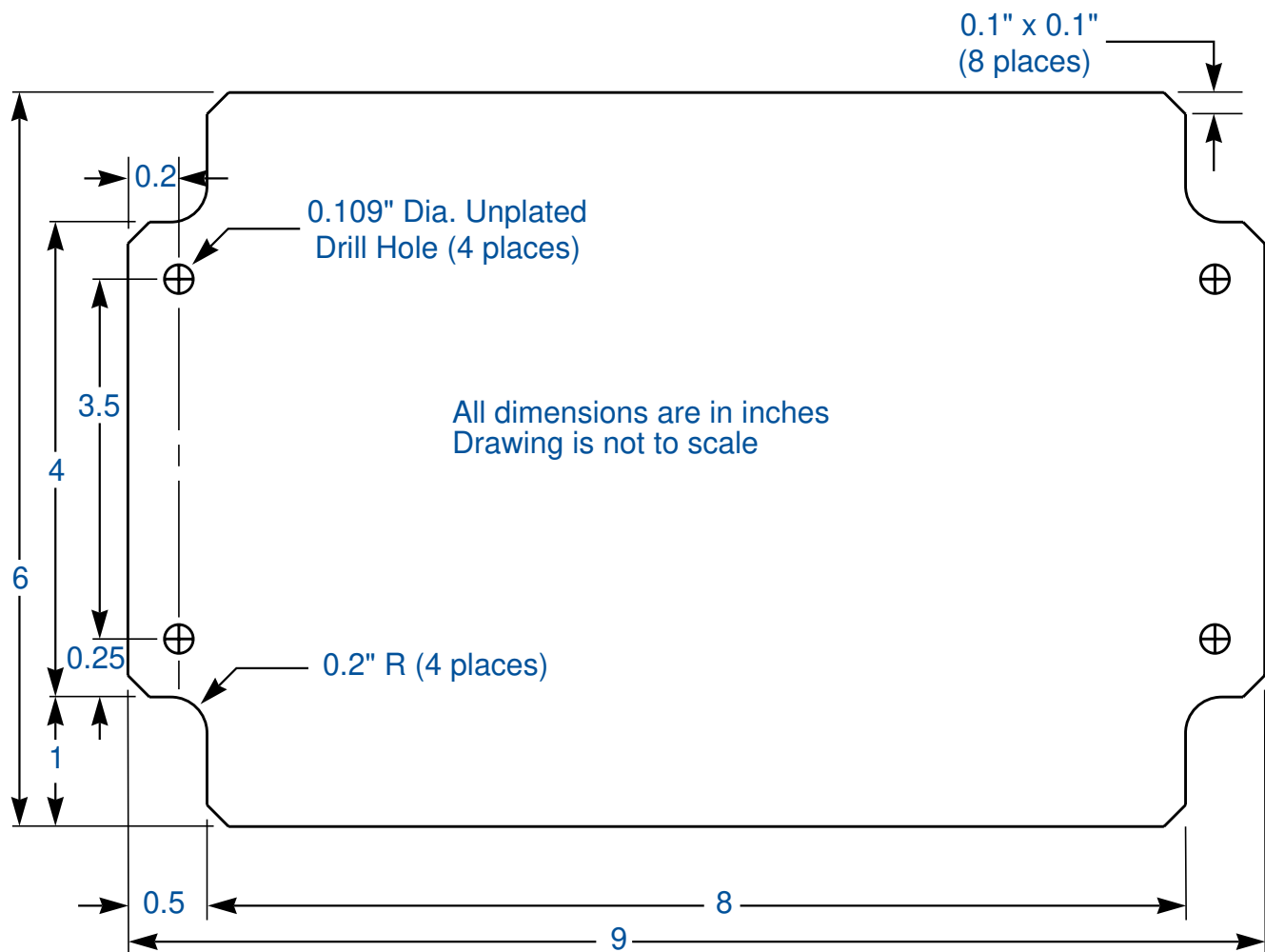


Figure 4-7. Outline for Board Geometry

Creating the Board Outline

In this procedure you specify the attributes of the board, open board geometry, and create the lines and arcs that form the board outline.

1. Choose the **Geometries > Create Geometry > Board...** menu item. Fill in the dialog box as follows, leaving all other options at the default values. When you finish, **OK** the dialog box.

Board Name: **signal_analyzer**
Board Material: **polyimide**
Board Thickness: **0.062**

Copper Thickness Internal: **0.0014**
External: **0.0014**

Default Padstack Name: **th055028**
Number of Routing Layers: **4**
Default Padstack Size: **0.055**
Route Power Nets: **To Power Layers**
Power Net Names: **vcc**
 +15v
 -15v
 ground

The board's default padstack (th055028) is used only when a component does not have a padstack already defined for it.

You entered the power net names in sequence. Later, you will set up split power planes, such that VCC and +15V are on a single physical layer, and -15V and ground share another physical layer. You must always enter power net names in sequence if you want them to later share a physical layer. All power net names for the board must be entered here.

2. Check that the edit layer is set to BOARD_OUTLINE, and the line width is set to 0.0. Set up the display grid to 0.05 with a display interval of 2. Make sure grid snapping is on.

3. Choose the **[Top Menu] Shapes** menu item to make Shapes the default popup menu.

Instead of constructing the outline of the board by working around its perimeter, you are going to use a more efficient method of constructing one end of the board first, and then you will copy it to the other end, and finally add horizontal lines to connect the two ends. In this way, you create only about half the geometry, because most of it you will copy. You will start by creating the left end of the board without fillets or chamfers, as shown in Figure 4-8.

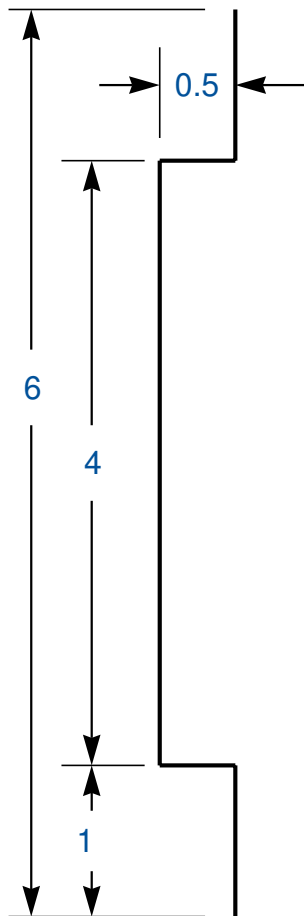


Figure 4-8. Beginning of Left End of Board

4. Choose the **[Shapes] Add Line > Add Line** menu item.

The Add Line prompt bar is visible. You are prompted for a series of locations through which the line will pass. The first point you specify is the Absolute coordinate location.

5. Choose the **[Shapes] Snap > Absolute** menu item. In the dialog box, enter **X= -4.5, Y= 3.0**, then **OK** the dialog box.

The Add Line prompt bar is visible again. Leave it there. It repeats for every point you enter.

You probably can't see the point you entered in the Edit window, because it is beyond the window's view area.

6. Place the cursor on the View Out icon, shown in Figure 4-9, and click the Select mouse button twice, or until you see the basepoint icon at the location you entered in the previous step.

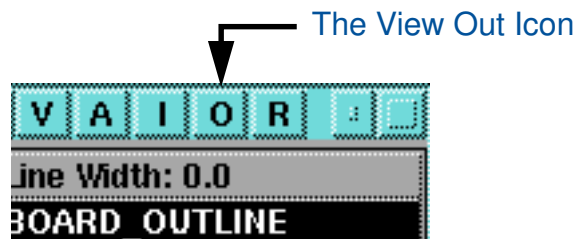


Figure 4-9. Locating the View Out Icon

The Add Line prompt bar is still visible. Now that you have entered the first point of the line, enter all remaining points in the Delta coordinate system.

7. Choose the **[Shapes] Snap > Delta** menu item. When the prompt bar is displayed, enter **X= 0, Y= -1, From Lastpoint**, and **OK** the prompt bar.

The location you entered is relative to the first point of the line, which was the last point you entered. In effect, you just entered the length of the line segment, which is 1 inch.

You might not see the line segment until you move the cursor.

The Add Line prompt bar is still visible.

8. Choose the **[Shapes] Snap > Delta** menu item. In the prompt bar, enter **X= -0.5, Y= 0, From Lastpoint**, and **OK** the prompt bar.

The Add Line prompt bar is still visible.

9. Choose the **[Shapes] Snap > Delta** menu item. In the prompt bar, enter **X= 0, Y= -4, From Lastpoint**, and **OK** the prompt bar.

The Add Line prompt bar is still visible.

10. Choose the **[Shapes] Snap > Delta** menu item. In the prompt bar, enter **X= 0.5, Y= 0, From Lastpoint**, and **OK** the prompt bar.

The Add Line prompt bar is still visible.

11. Choose the **[Shapes] Snap > Delta** menu item. In the prompt bar, enter **X= 0, Y= -1, From Lastpoint**, and **OK** the prompt bar.

The Add Line prompt bar is still visible.

12. In the Add Line prompt bar, click on **OK**, and then when the Add Line prompt bar repeats again, click on **Cancel**.

The rough outline of the left end of the board is done and appear as shown in Figure 4-10. Add fillets and chamfers before you copy the left end to make the right end.

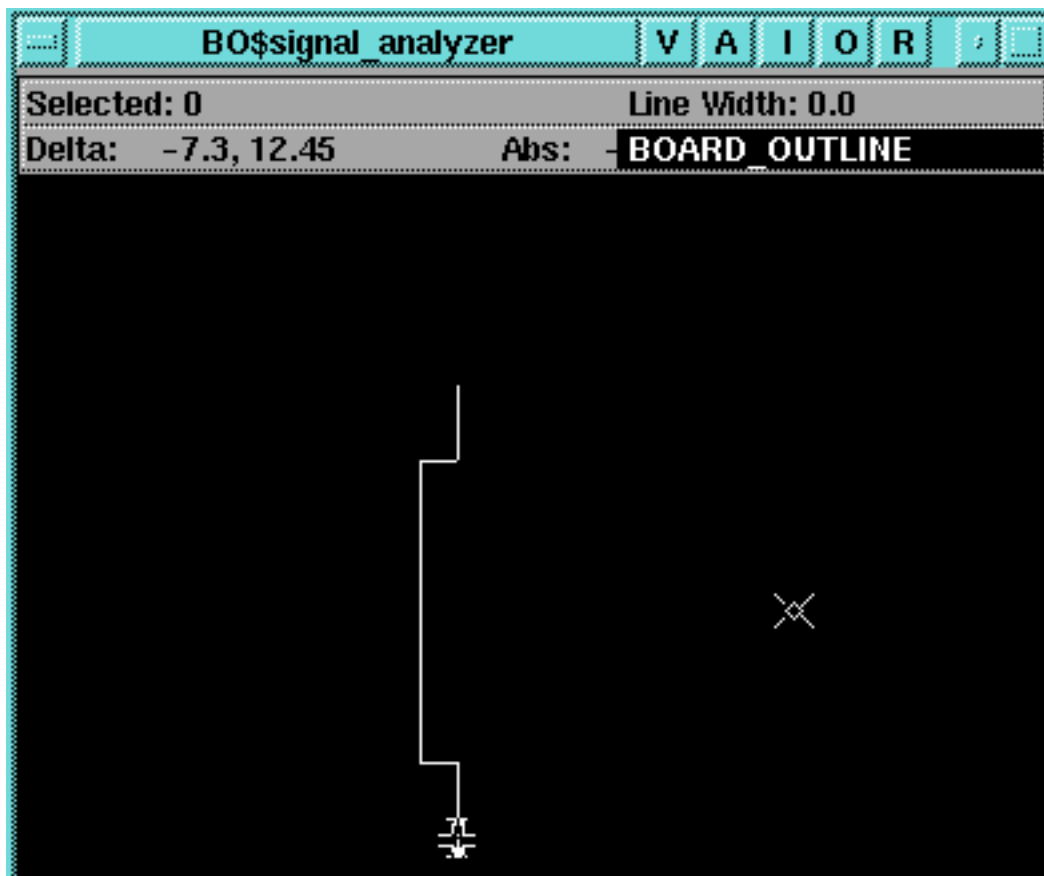


Figure 4-10. Completed Rough Left End

You had to OK the Add Line prompt bar to complete the line. If you had clicked on Cancel before OK, the line would have been removed.

13. Cancel any prompt bars that are visible, if any.

Right now, the line you created is a single entity, which means if you select it, the entire line will be selected as one object. Before you can add chamfers or fillets and have the ends of the lines automatically trimmed back to the chamfers or fillets, you must break the line into separate line segments.

14. Place the cursor on the line, and click the Select mouse button to select the line.

One object is selected.

15. Choose the **[Shapes] Change > Entity Type** menu item. In the dialog box, choose **Segment**, and then **OK** the dialog box. Finally, unselect all objects (5) by pressing the Unselect All function key.
16. Zoom in to view the upper half of the line you have created, so you have a clear view of the corner where the chamfer will be placed.

Now you are going to place the two chamfers on the two corners of the left end.

17. Choose the **[Shapes] Add Line > Chamfer** menu item. In the dialog box, enter Angle: **45**, Distance **0.1**. Choose **clip_both** for the Clip Option. Finally, TAB to the From prompt. Place the cursor on the location shown in Figure 4-11 for the *From* point, and click the Select mouse button. At the To prompt, place the cursor on the location shown in Figure 4-11 for the *To* point, and click the Select mouse button.

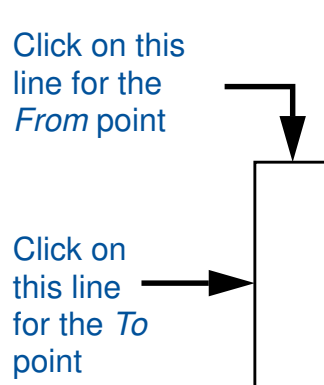


Figure 4-11. Locations for First Chamfer

The Add Chamfer dialog box is repeated so you can add the second chamfer.

18. Change your view of the line so you see its lower end, where the other chamfer will go.

19. Place another chamfer at the lower corner, as you added the first one. The chamfer is **45** degrees, **0.1** inches, and you must use the **clip_both** option.

20. Cancel any prompt bars.

Now you will add the two fillets, for which you use a similar technique as for adding the chamfers.

21. Choose the **[Shapes] Add Arc > Make Fillet** menu item. In the dialog box, enter a radius of **0.2**, and make sure the Clip Option is set to **clip_both**. Press the TAB key to highlight the From prompt. Place the cursor on the location shown in Figure 4-12 for the *From* point, and click the Select mouse button. At the To prompt, place the cursor on the location shown in Figure 4-12 for the *To* point, and click the Select mouse button.

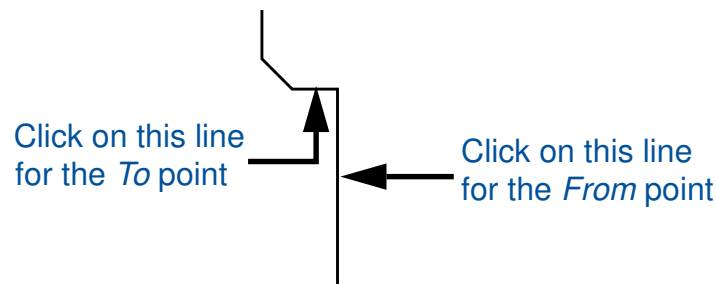


Figure 4-12. Locations for First Fillet

The first fillet is added, and the Add Fillet prompt bar is repeated so you can add the other fillet.

22. Add the second fillet at the corner near the first chamfer. Use the same settings as for the first fillet.

Now that the left end of the board is complete, as shown at the left side of Figure 4-13, you will copy it to the right end. First you will select the left end you have made, then copy it 8 inches in the X direction, relative to the basepoint. Then you will flip the right end copy so it will be correctly positioned. Finally, you will join the two ends and add the final four chamfers.

23. Cancel all prompt bars.

24. View all of the geometry, and then select all of it.

There are 9 objects selected.

To select all objects in an area, place the cursor above and to the left of the top of the geometry, hold down the Select mouse button, drag the cursor down and to the right until the selection rectangle completely encloses the geometry, and release the Select mouse button.

25. Choose the **[Shapes] Copy** menu item. When the copy prompt bar is displayed, choose the **[Shapes] Snap > Delta** menu item. In the prompt bar, enter X= **8**, Y= **0**, From **Basepoint**, then **OK** the prompt bar.

The copy is made, but you can't see it because it is out of the view area.

26. Cancel the Copy prompt bar that repeated, then view all of the geometry. It appears as shown in Figure 4-13, and the copy at the right end is still selected.

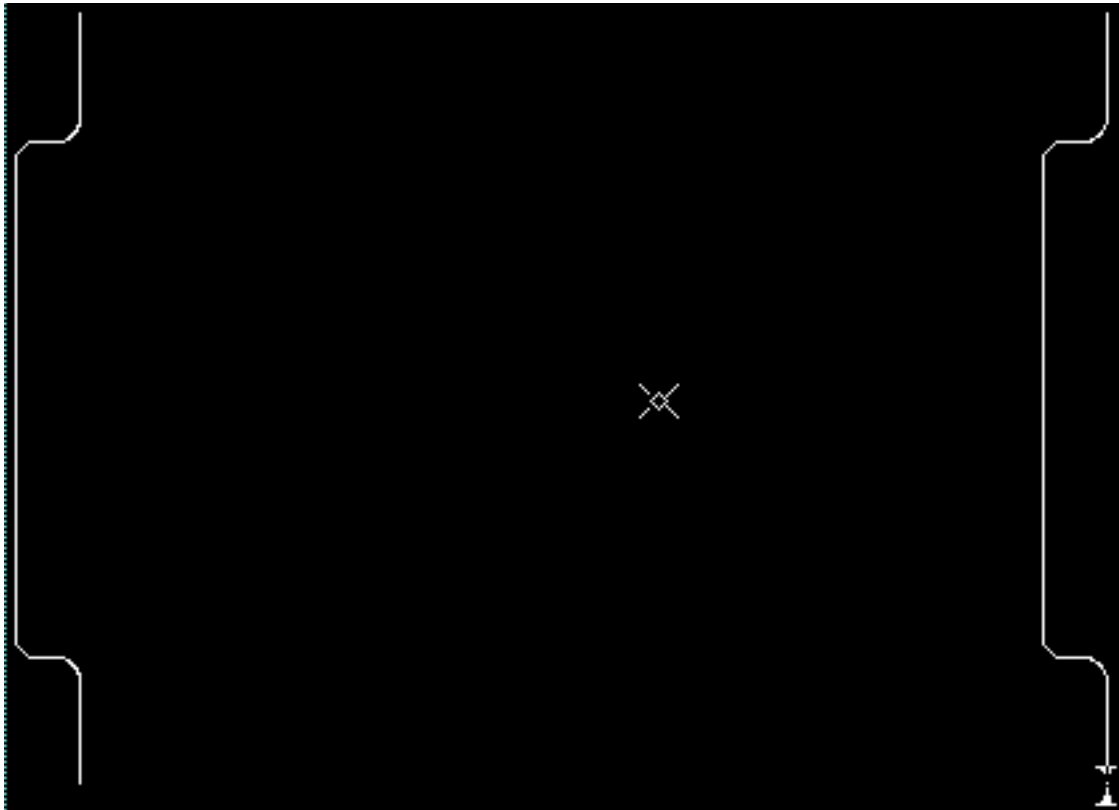


Figure 4-13. The Left End Copied to the Right

Before you flip the right end to correctly orient it, you need to move the basepoint so it is located as shown above in Figure 4-13.

27. Make sure only the right-end geometry (the copy you made) is selected (only 9 objects). If it is not selected, select it.
28. Choose the **[Shapes] Move > Basepoint** menu item. Next, choose the **[Shapes] Snap > Endpoint**. Place the cursor on the lowest end of the right side copy, and click the Select mouse button.

The basepoint is moved so it is located as shown in Figure 4-13, and the right-side copy is still selected.

29. Choose the **[Shapes] Flip > Horizontally** menu item.

The right side is now oriented so it appears as in Figure 4-14.

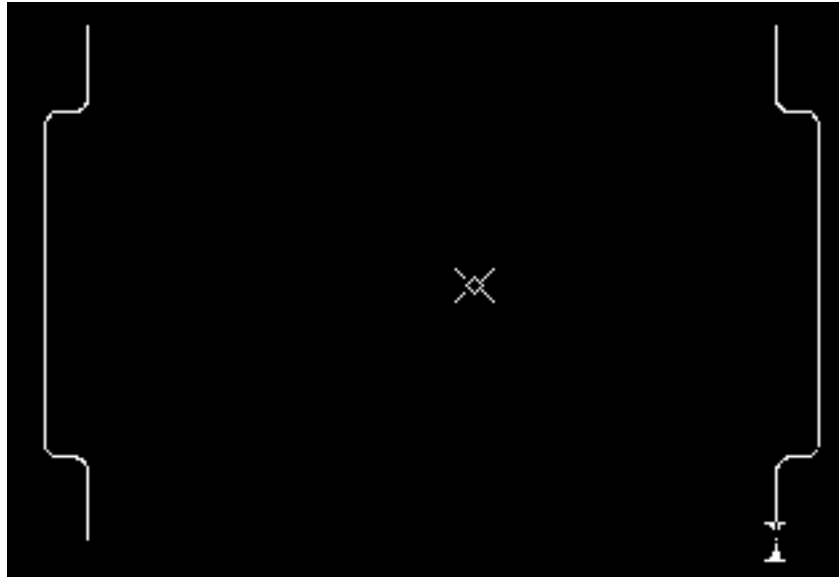


Figure 4-14. The Right Side After Flipping

30. Unselect all objects, and cancel any prompt bars, if any.
31. Choose the **[Shapes] Add Line > Add Line** menu item. Then choose the **[Shapes] Snap > Endpoint** menu item. Place the cursor on the line at the top of the left end, and click the Select mouse button. Choose the **[Shapes] Snap > Endpoint** menu item again, place the cursor on the line at the top of the right end, and click the Select mouse button. Finally, **OK** the Add Line prompt bar.

The line connecting the top of the two ends is completed. The Add Line prompt bar is repeated so you can create another line.

32. Choose the **[Shapes] Snap > Endpoint** menu item. Place the cursor on the line at the bottom of the left end and click the Select mouse button. Choose the **[Shapes] Snap > Endpoint** menu item again, place the cursor on the line at the bottom of the right end, and click the Select mouse button. **OK** the Add Line prompt bar.

The line connecting the bottom of the two ends is completed. The Add Line prompt bar is repeated.

33. Cancel the Add Line prompt bar.
34. Add the chamfers to the four new corners of the board, using the method you used in step 17.

The finished outline appears as shown in Figure 4-15.

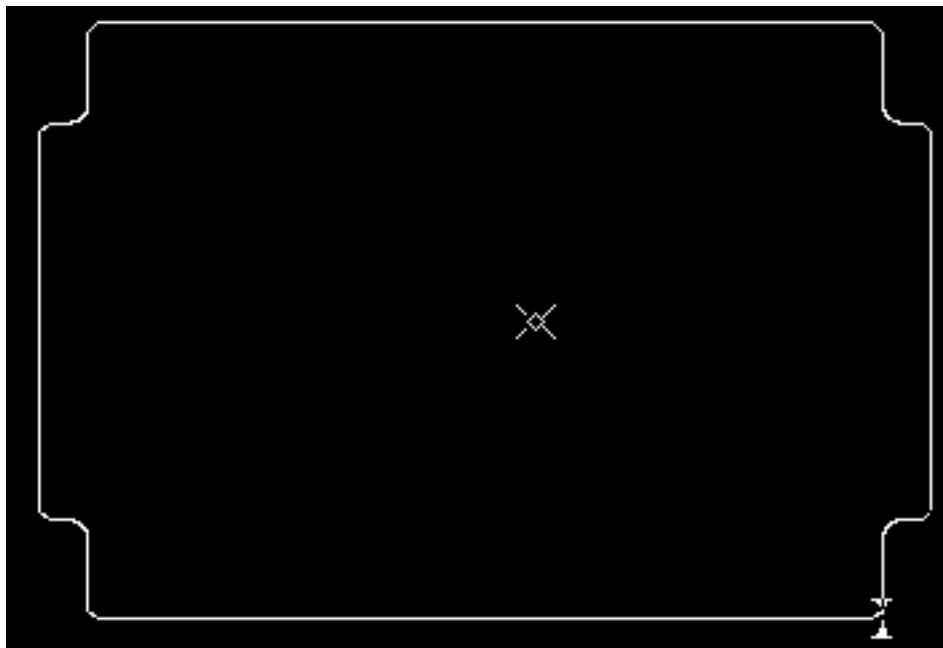


Figure 4-15. The Complete Outline

Now the board outline looks complete. However, it is just a set of individual lines and arcs. To complete the outline, you must make all the segments into a single item. To do this, you first set up the select filter to only select lines and arcs. Then you use the select all menu item to select all segments of the board outline and join all the segments into a polygon.

35. View all of the board geometry, using the View All stroke.
36. Choose the **Setup > Select Filter** menu item. In the Setup Select Filter dialog box, first choose **Clear All** to unhighlight all options. Then choose **Arcs** and **Lines** only, and **OK** the dialog box.

From now on, until you change the select filter in this session, the Select All menu item will select only lines and arcs.

37. Choose the **[Shapes] Select > Select All** menu item.
38. Choose the **[Shapes] Change > Entity Type...** menu item. In the dialog box that appears choose **Path**. Enter **8** for Number of edges (for circles/arcs). **OK** the dialog box.

The board outline graphics is combined into a single continuous path element.

39. If the board did not remain selected, select it again.

There is only one item selected because you have joined all the pieces together. Now you are going to move the board so its lower-left corner is at the geometries origin.

40. Choose the **[Shapes] Move > Origin** menu item. Then choose the **[Shapes] Snap > Intersection** menu item. Next, click the Select mouse button on the lower horizontal edge of the board, and then on the lower-left vertical edge of the board. Refer to Figure 4-16.

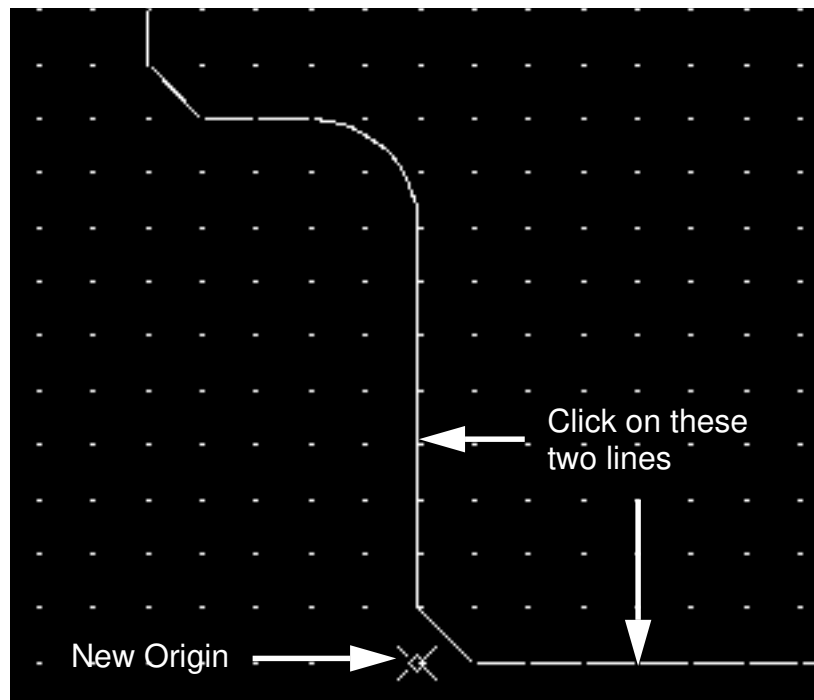


Figure 4-16. Relocating the Origin

The intersection of the left edge of the board and the bottom edge of the board is at absolute coordinate $x=0$, $y=0$; the origin of the geometry.

41. Unselect all items by pressing the Unselect All function key.

Adding the Drill Holes

You are ready to add the mounting holes. When you add the holes, you are going to specify a location in the Delta coordinate system. You will enter the specifications for the drill hole first. To locate the drill hole, you first place the basepoint at the intersection near where the drill hole will be placed, then you will specify a coordinate relative to the basepoint location. You will add the first drill hole at the lower corner of the left end of the board.

Refer to Figure 4-7 at the beginning of this lab for the locations of all the drill holes. All the drill holes are the same distance away from their respective corners.

1. View all of the board geometry using the View All stroke.
2. Choose the **[Shapes] Extended Menu > Add Drill Hole** menu item. In the prompt bar, enter **.109** for the Diameter. Choose **Options**, and in the options dialog box, choose Drill Hole Type **Unplated**. **OK** the dialog box. In the prompt bar, Tab to the location prompt.
3. Choose the **[Shapes] Move > Basepoint** menu item. Next, choose the **[Shapes] Snap > Intersection** menu item. Click the Select mouse button once on each of the two lines shown in Figure 4-17.

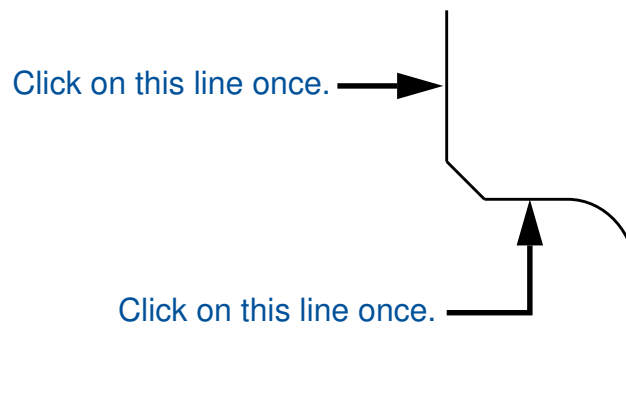


Figure 4-17. Locating the Basepoint for the First Drill Hole

You might not see the basepoint at the new location until you move the cursor a little. Although the two lines you clicked on do not

actually intersect, LIBRARIAN locates the projected intersection of those lines and places the basepoint there.

Now that you have a basepoint conveniently located, you can specify a coordinate relative to it.

The Add Drill Hole prompt bar is still visible, and you are still being prompted for a location.

4. Choose the **[Shapes] Snap > Delta** menu item. In the prompt bar, specify X= **0.2**, Y= **0.25**, and set the From prompt to **basepoint**. **OK** the prompt bar.

When you move the cursor into the edit window, the drill hole is displayed. The Add Drill Hole prompt bar repeats.

Now you will add the drill hole at the upper corner of the left end.

5. Tab to the location prompt in the Add Drill Hole prompt bar, then move the basepoint to the intersection near where the drill hole at the upper corner of the left end will be located. Finally, enter the coordinates for the drill hole from the basepoint. Follow the procedure you used in placing the first drill hole.

Instead of creating the second drill hole, you could copy the first one to the second one's location.

6. Add the two other drill holes at the right end of the board, as shown in Figure 4-7. When you are done placing drill holes, cancel the Add Drill Hole prompt bar.

Optionally, you could select the first two drill holes and copy both of them at once to the right end of the board.

Adding the Placement Outline

Next you will add the placement outline to the board. This defines the area in which it is legal to place components.

When you add the placement outline, refer to Figure 4-18 for views of the completed placement outline, and a detailed view near a corner of the board.

1. Choose the **[Shapes] Return to Top Menu** menu item. Next, choose the **[Top Menu] Attributes** menu item.

Now the Attributes menu is the default popup menu.

2. Choose the **[Attributes] Create Placement Outline** menu item. In the dialog box that is displayed, choose **Interactive**, then **OK** the dialog box. When the prompt bar prompts you for locations, place the cursor where you want a vertex (corner) of the outline to be (refer to Figure 4-18) and click the Select mouse button. Move the cursor and click the Select mouse button at each vertex around the polygon, going all the way around the board. Keep the placement outline about 0.2 inch in from the edge of the board (two grid spaces), except at the left and right board edge, where the placement outline lies exactly on the board edge. You must leave enough room between the placement outline and the board edge so you can add the routing outline in the next step. Press the RETURN key to complete the prompt bar.

The placement outline must lie directly on the left and right board edge so that the connectors can be placed as close as possible to the board edge.

If you misplace a vertex, and you want to back up the outline to the previous vertex to repair the outline, press the Backspace key on the keyboard. The Backspace key removes the previous vertex so you can reposition it. You can remove several vertices at once by pressing the Backspace key for each vertex you want to remove.

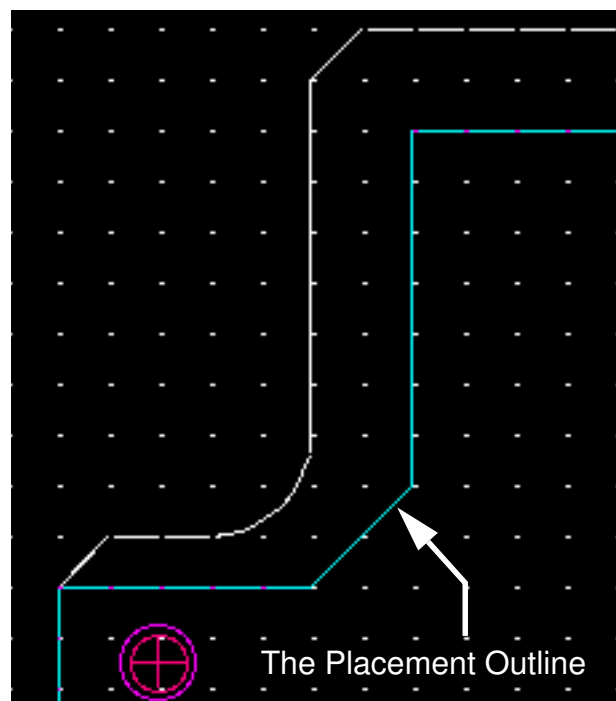
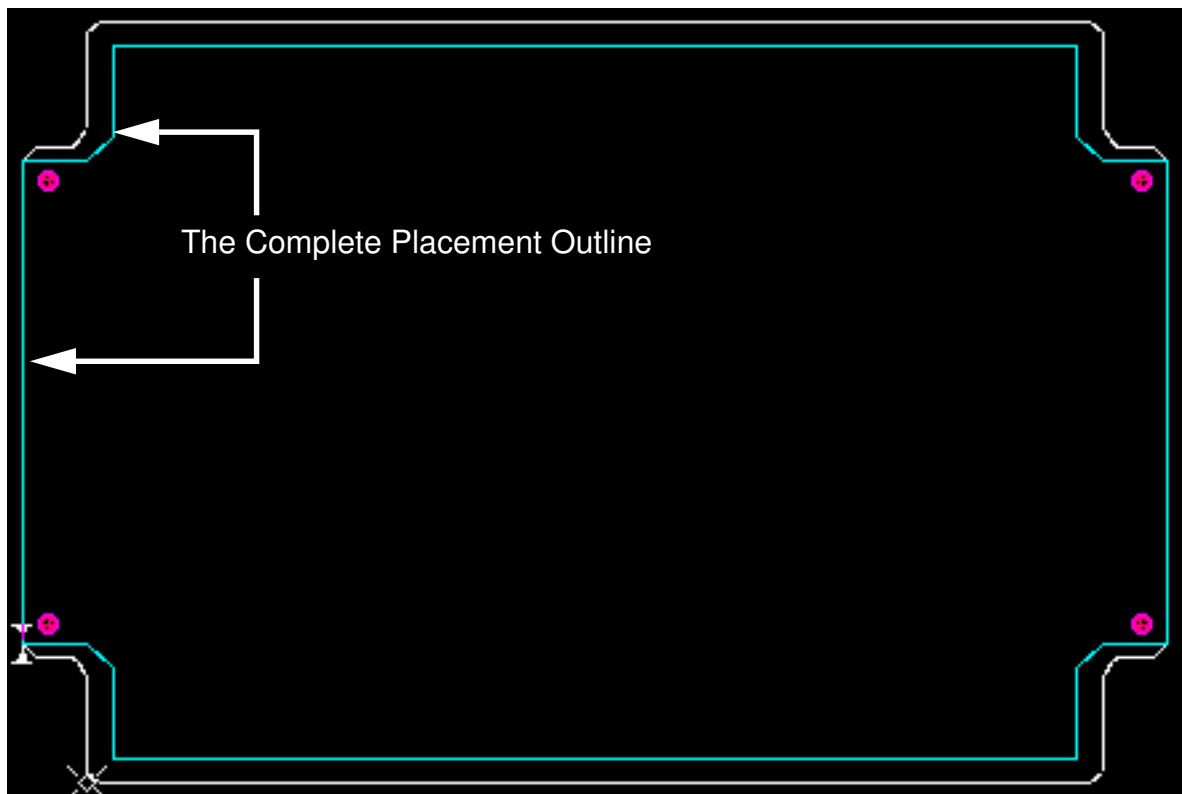


Figure 4-18. Details of Placement Outline

Adding the Routing Outline

Now you add the routing outline to the board. You will interactively place the routing outline between the placement outline and the edge of the board. This is necessary so that if a component is placed at the extreme edge of the placement outline, there will be room between the component and the edge of the routing outline to route traces. You do not use the automatic feature to add the routing outline, because it automatically keeps the outline a minimum distance between the outline and the board edge all the way around the board. You need to keep the routing outline right at the board edge on the left and right edges to leave enough routing room for the connector pins.

At the left and right edges of the board, you will place the routing outline exactly on the edge of the board, as you did when you added the placement outline.

While you are adding the routing outline, refer to Figure 4-19.

1. Choose the **[Attributes] Create Routing Outline** menu item. Use the Select mouse button to add the routing outline as shown in Figure 4-19. Keep the routing outline about .1 inch in from the edge of the board (one grid space). Notice in Figure 4-19 how the routing outline goes around the mounting holes. Press the RETURN key to complete the prompt bar.

The mounting holes are for mounting the 96 pin connector you made earlier. The holes are spaced identically to the mounting holes on the connector. The placement outline goes between the mounting holes and the edge of the board so the connector can be placed inside the placement outline. The routing outline is positioned inside of the holes, because that is where the pins will be for the connectors.

As when you added the placement outline, click the Select mouse button once at each vertex. You can use the Backspace key on the keyboard to remove one or more vertices if you make a mistake.

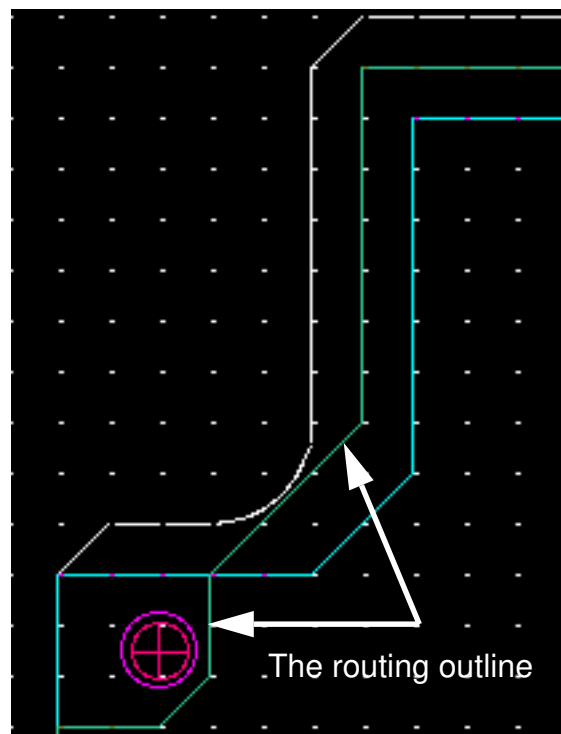
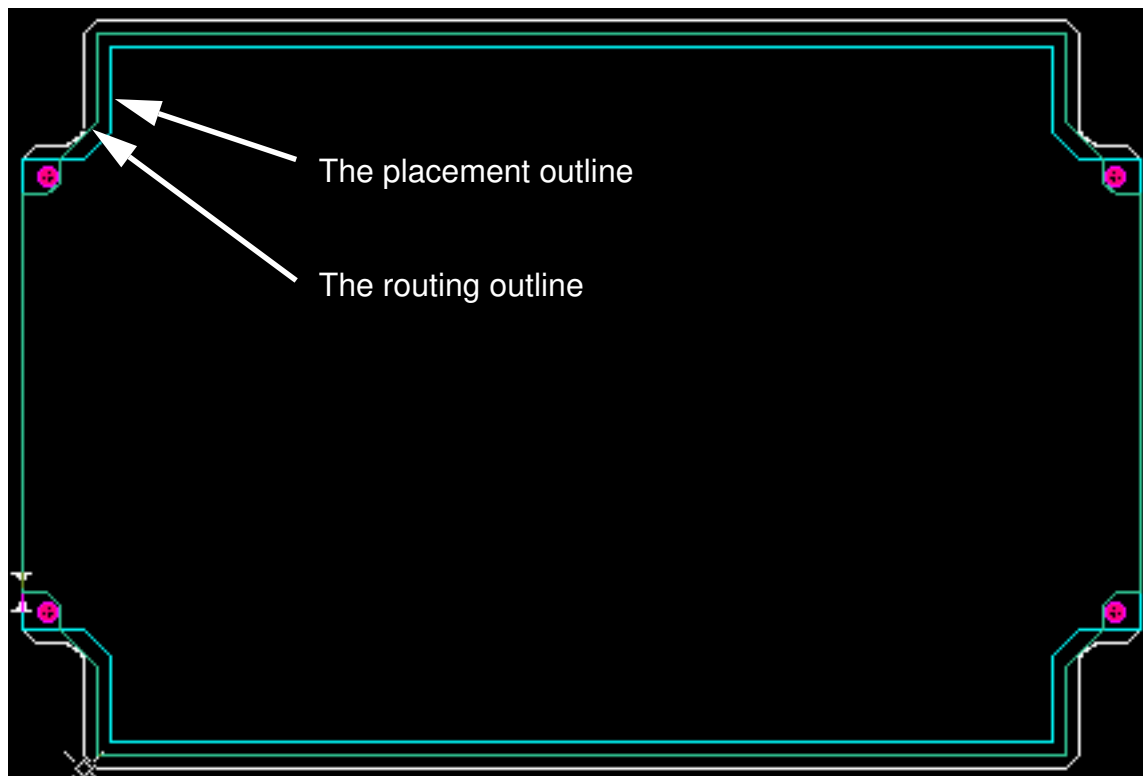


Figure 4-19. Details of Routing Outline

Create a Library Link to the trng Library



If you are taking this training course as part of a workshop, skip this section, and begin with section "Reading the conn_DIN96 Geometry" on page 4-39. Create the link to geometry library only if you are completing this training as a Personal Learning Program.

In the workshop version of this training material, the libraries are set up for you by your instructor.

In the next procedure, you read the conn_DIN96 geometry into the LIBRARIAN session. These and the other geometries you created in the previous lab you saved in a library named *trng*, which you created. Before you can read the conn_DIN96 geometry into the LIBRARIAN session, you must create a library link to the *trng* library. In this procedure you create this library link, just as you created library links to the other libraries in a previous lab session. After you create the library link, you will read the conn_DIN96 geometry before you add it to the board.

1. Choose the **Geometries > Add Library Link...** menu item, enter the following in the dialog box, and then **OK** the dialog box.

Library Name: **mgc.trng**

Pathname to Existing Library:

your_path/training/board_new/mod3/sig_az/pcb_parts/
user_geom/trng

Add to: **User**

Directory type: **Permanent**

Reading the conn_DIN96 Geometry

In this procedure, you read the conn_DIN96 geometry from your User parts library.

1. Choose the **Geometries > List Geometry Libraries...** menu item. In the dialog box, from the list of libraries, select User library. Then, select **mgc.trng**. Move the cursor to the bottom of the dialog box and select the **View** button. Select the **conn_DIN96** geometry, then select the **Read** button at the bottom of the dialog box.

The conn_DIN96 geometry opens in an Edit window on top of the board geometry Edit window.

2. From the Window menu, choose **Pop** to push the conn_DIN96 Edit window under the board geometry Edit window.
3. Return to the [Top Menu] popup menu, then choose **[Top Menu] Shapes** menu item to make Shapes the default popup menu.
4. Add the conn_DIN96 to the board by choosing the **[Shapes] > Extended Menu > Fix Component** menu item. Fill in the prompt bar as follows, then TAB to the Location prompt:

Reference: **conn_1**
Geometry: **conn_DIN96**
Top
Rotation: **d270**

5. Choose the **[Shapes] Snap > Center** menu item, then place the cursor on the lower-left drill hole, and click the Select mouse button.

When you move the cursor in the Edit window, the connector is added to the left end of the board. Because you moved the origin of the connector geometry to the connector's drill hole, the connector's drill hole was placed at the center of the board's drill hole. An example of the board outline with the connector placed is shown in Figure 4-20.



Figure 4-20. Board Outline with the Left Connector

Next you add the connector on the right end of the board. The Fix Component prompt bar is already visible at the bottom of the screen.

6. Fill in the Fix Component prompt bar as follows, then Tab to the location prompt.

Reference: **conn_2**
Geometry: **conn_DIN96**
Top
Rotation: **d90**

7. With the location prompt highlighted, choose the **[Shapes] Snap > Center** menu item. Place the cursor on the upper-right mounting hole, and click the Select mouse button.

Refer to Figure 4-21 for an example of the board outline geometry with both connectors.



Figure 4-21. Board Outline with Both Connectors

The Fix Component prompt bar is repeated.

8. Cancel all prompt bars.

If you make a mistake placing the components, you can either choose the **[Shapes] Geometric Undo** menu item, or you can choose the **[Shapes] Extended Menu > Unfix Component** menu item, and then select the components reference name from the dialog box. This removes the connector. You can then add the connector again, using a different location. A fixed component cannot be selected and moved like other geometry.

With the physical part of the board complete, it is now time to set up some of the design rules.

Setting Default Design Rules

The first step in setting up design rules is to check the physical layering of the board and make required changes. Currently, the board has four signal layers and four power layers. You are going to arrange them so that there is a signal layer on top, followed by two power layers, followed by two signal layers, followed by two more power layers, with the final signal layer being on the very bottom. You will also rename all the layers so they are easier for you to recognize, and you will specify that the board is double sided.

When you are finished changing the physical rules, you will specify the default vias, and define the via rules.

Defining Physical Layers

1. Choose the **Setup Design Rules > Physical Layers...** menu item. Place the cursor on physical layer **PHYSICAL_2** and click the Select mouse button. Press the **Delete...** button at the bottom of the dialog box. In the confirmation dialog box that is displayed, choose **Yes**.

Physical layer 2 is removed.

2. Using the same method, select and delete physical layer **PHYSICAL_7**.
3. Select physical layer **PHYSICAL_5**, and then press the **Insert New...** button at the bottom of the dialog box. In the Insert New Physical Layer Definition dialog box that is displayed, enter the following and then **OK** the Insert New Physical Layer Definition dialog box.

Physical Layer Name: **Trace_layer_2**

Logical Layer Name: **SIGNAL_2**

4. Select physical layer **PHYSICAL_5**, and then press the **Insert New...** button at the bottom of the dialog box. In the Insert New Physical Layer Definition dialog box that is displayed, enter the

following and then **OK** the Insert New Physical Layer Definition dialog box.

Physical Layer Name: **Trace_layer_3**

Logical Layer Name: **SIGNAL_3**

You have now redefined how the board is laminated together. Now, look at the power plane layers, Physical_3, Physical_4, Physical_5, and Physical_6. Those names are not very descriptive. Next, you will change the names to something better.

5. In the Setup Physical Layers dialog box, select **PHYSICAL_3 (POWER_1)**, and then press the **Change...** button. In the Change Physical Layer Name dialog box, enter the following, and then **OK** the Change Physical Layer Name dialog box.

Physical Layer: **VCC**

6. In the Setup Physical Layer dialog box, select **PHYSICAL_4**, and then press the **Change...** button. In the Change Physical Layer Name dialog box, enter the following, and then **OK** the Change Physical Layer Name dialog box.

Physical Layer: **POS15V**

7. Change the name of PHYSICAL_5 to NEG15V, and PHYSICAL_6 to ground. Make sure *ground* is in lowercase.
8. Change the name of PHYSICAL_1 to Trace_layer_1, and PHYSICAL_8 to Trace_layer_4.
9. Click on the **Design_Info...** button at the bottom of the Setup Physical Layers dialog box. In the Design Information dialog box that is displayed, choose **Double Side**, and then **OK** the dialog box.
10. Verify that the contents of the Setup Physical Layers dialog box matches Figure 4-22.

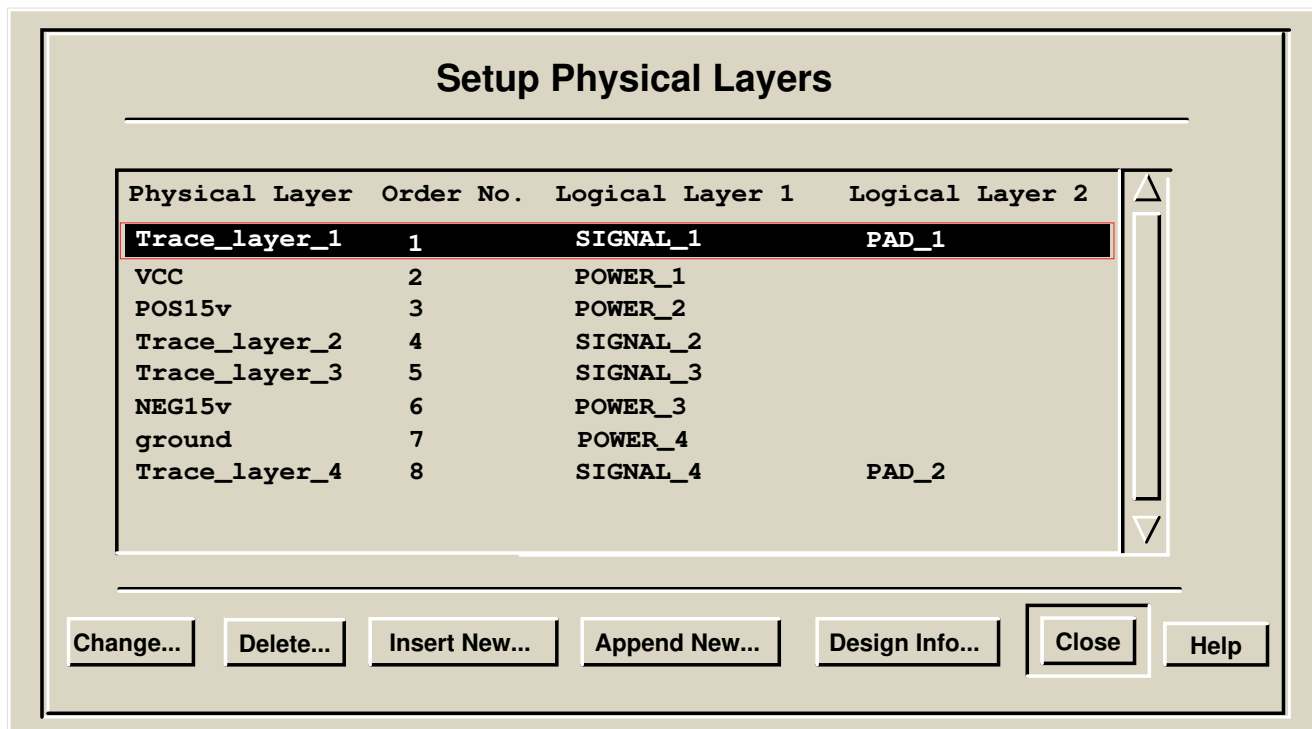


Figure 4-22. Final Contents of the Setup Physical Layers Dialog Box

11. Close the Setup Physical Layers dialog box.

Defining Layer Rules

Now, set up the rules defining routing constraints on the trace layers.

1. Choose the **Setup Design Rules > Layer Rules...** menu item. Select **Trace_layer_2**, and change the preferred direction to **Vertical**. Select **Trace_layer_3**, and change the preferred direction to **Horizontal**. Finally, **Close** the Setup Layer Rules dialog box.

The Vertical and Horizontal settings specify the primary direction for routing relative to the display screen.

Because you are setting up the design rules for this design, it is appropriate for you to choose the vias to use for routing. In the next step, you will read the vias you need from the User parts libraries.

2. Choose the **Geometries > List Geometry Libraries** menu item. In the dialog box, click on **User** libraries to select it. In the list of user libraries that is displayed, select **mgc.trng.padstacks**, and then press **View**. Select the **via040015**, **power_via_front**, and **power_via_back** padstacks, and then press **Read**.

The three via geometries are read from the ASCII libraries into the session. You see them as edit windows.

Defining Via Rules

Now, set up the rules for the vias for the design.

Setting up Via040015

1. Choose the **Setup Design Rules > Via Rules** menu item. In the Change Via Rules dialog box that is displayed, you see the three vias that you just read. Select **via040015** from the list of via names, and then click on the **Setup Rules...** button.
2. Choose the **Add...** button. In the Add Via Rules dialog box that is displayed, enter the following, and **OK** to return to the Setup Buried or Two-layer Via Rules dialog box:

Start Layer: Trace_layer_1	Stop Layer: Trace_layer_2
Start Layer: Trace_layer_2	Stop Layer: Trace_layer_3
Start Layer: Trace_layer_3	Stop Layer: Trace_layer_4

3. In the Rules for Via Under Pad list, select **Trace_layer_1**, and then click on the **Connect...** button. In the Connect Via Rules dialog box that is displayed, enter the following, then **OK** the Connect Via Rules dialog box so you return to the Setup Buried or Two-layer Via Rules dialog box:

Specify Layers

Layer Names: **Trace_layer_1**
Trace_layer_4

The **Connect...** button allows a via to be coincident with a pin on the layers you specify. On **Trace_layer_1** and **Trace_layer_4** you do not need to create a trace between a pin and a *via040015* via. The autorouter can also add this via directly on a pin on those layers.

In the Setup Buried or Two-layer Via Rules dialog box, under the Rules for Via Under Pad list, **Trace_layer_1** and **Trace_layer_4** show as Connection Enabled, and all other layers are Disabled.

The dialog box is now filled in as shown in Figure 4-23.

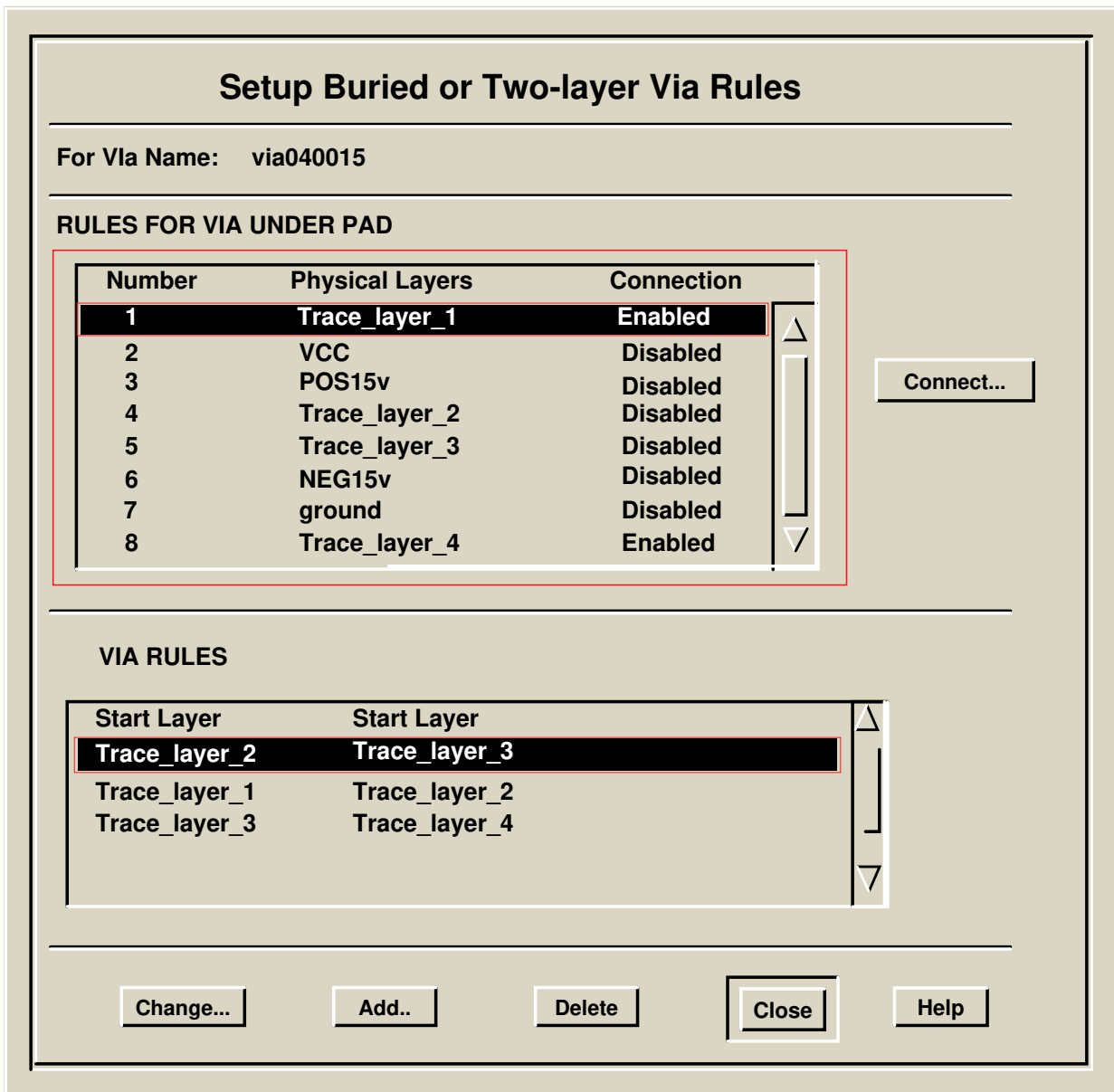


Figure 4-23. Completed Via Rules for via040015

4. After you verify the entries in the dialog box, choose **Close**.

Setting up Power_via_front

1. Choose the **Setup Design Rules > Via Rules** menu item again. Select **power_via_front** from the list of via names, and then click on the **Setup Rules...** button.
2. In the Setup Buried or Two-layer Via Rules dialog box, Enable **Trace_layer_1**. All other layers are Disabled.
3. Now choose the **Add...** button. In the Add Via Rules dialog box that is displayed, enter the following, and **OK** to return to the Setup Buried or Two-layer Via Rules dialog box:

Start Layer: Trace_layer_1	Stop Layer: VCC
Start Layer: Trace_layer_1	Stop Layer: POS15V
Start Layer: Trace_layer_1	Stop Layer: NEG15V
Start Layer: Trace_layer_1	Stop Layer: ground

You cannot specify start and stop layer names that begin with the minus sign -, or with the plus sign +, which is why you named the physical layers POS15V and NEG15V. The names of the physical layers do not have to match anything.

4. Close the Setup Buried or Two-layer Via Rules dialog box.

Setting up Power_via_back

1. Choose the **Setup Design Rules > Via Rules** menu item again. Select **power_via_back** from the list of via names, and then click on the **Setup Rules...** button. Next, choose the **Connect** button.
2. In the Connect Via Rules dialog box that is displayed, choose **Specify Layers**, and then enter **Trace_layer_4** in the Layer Names field. **OK** the Connect Via Rules dialog box.

Trace_layer_4 displays as Connection Enabled, and all other layers are Disabled.

3. Now choose the **Add...** button. In the Add Via Rules dialog box that is displayed, enter the following information and select **OK** to return to the Setup Buried or Two-layer Via Rules dialog box:

Start Layer: VCC	Stop Layer: Trace_layer_4
Start Layer: POS15V	Stop Layer: Trace_layer_4
Start Layer: NEG15V	Stop Layer: Trace_layer_4
Start Layer: ground	Stop Layer: Trace_layer_4

4. Close the Setup Buried or Two-layer Via Rules dialog box.

Closing this dialog box saves the entries you made. Now you set up the Net Rules design rules for the board.

Defining Net Rules

In this section, you set up rules governing the routing of nets grouped by net type. The first net type is the default. Rules defined in the default net type are applied to all nets that do not have a Net_type property attached. You can have as many net types as you need. You give each net type whatever value has meaning to you.

1. Choose the **Setup Design Rules > Net Rules...** menu item. In the dialog box, choose **Add...**

The Setup Net Rules dialog box displays.

2. In the Setup Net Rules dialog box, enter the following and then **OK** the dialog box.

Net Type **DEFAULT_NET_TYPE**
Trace Width **.01**

Pin to Pin Clearance **.01**
Via to Via Clearance **.01**
Pin to Trace Clearance **.01**
Via to Trace Clearance **.01**
Pin to Via Clearance **.01**
Trace to Trace Clearance **.01**

Select Interactive Routing Routing Vias: **via040015**

Choose one or more of the SELECTED Interactive Routing Vias for Auto Routing: **via040015**

Available Routing Layers: **Trace_layer_1**
Trace_layer_2
Trace_layer_3
Trace_layer_4

Force Changes even if these rules are in use: (**make sure this option is turned off (unhighlighted)**)



To toggle multiple layers on or off individually, you press the CTRL key while clicking the Select mouse button on the desired layer. For example, to select Trace_layer_2 and Trace_layer_3 at the same time you drag the select mouse button over both of them. You then select Trace_layer_1 by placing the cursor on it, holding down the CTRL key, and clicking the Select mouse button. You can select Trace_layer_4 the same way.

3. In the Setup Net Rules dialog box, choose **Add...**

4. In the Setup Net Rules dialog box, enter the following and then **OK** the dialog box.

Net Type **POWER_NETS**

Trace Width **.025**

Pin to Pin Clearance **.01**

Via to Via Clearance **.01**

Pin to Trace Clearance **.01**

Via to Trace Clearance **.01**

Pin to Via Clearance **.01**

Trace to Trace Clearance **.01**

Select Interactive Routing Routing Vias: **power_via_front**
power_via_back

Choose one or more of the SELECTED Interactive Routing Vias
for Auto Routing: **power_via_front**
power_via_back

Available Routing Layers: **Trace_layer_1**
Trace_layer_2
Trace_layer_3
Trace_layer_4

5. In the Setup Net Rules dialog box, choose **Close**.

Checking the Board Geometry

Now, you check the board geometry for errors.

1. Choose the **Check > Geometry > Geometries** menu item. In the Check Geometries dialog box, choose **All**, then **OK** the dialog box.

A report window is displayed. You might see a warning about no clearances, but this is okay. However, you will also see two or more errors that result from missing padstacks. Remember, at the beginning of this lab, when you filled in the Create Board dialog box you specified that the default padstack for the board is the th055028 padstack. Also, the connector conn_DIN96 requires the th055028 and th055028sq padstacks. Before you again check the geometry, you must read these padstacks into the LIBRARIAN session.

2. Close the report window.
3. Choose the **Geometries > List Geometry Libraries** menu item. In the dialog box, view the User libraries, and then view the contents of the padstacks library. From the padstacks library, read the th055028 and th055028sq padstacks.

These padstacks are read into the session.

4. Choose the **Check > Geometry > Geometries** menu item. In the Check Geometries dialog box, choose **All**, then **OK** the dialog box. In the report window that is displayed, make sure there are no errors.

You might see a warning about no clearances, and other warnings about Drill Definition Unplated, but this is okay. If you find any errors, however, you must correct them.

Saving Geometries and Leaving Librarian

Now that you have created the board geometry, you need to save it in ASCII format in your `mgc.trng` User library where you saved all your other geometries. Later, you will save another binary copy as part of the design `sig_az`. The ASCII copy serves as a backup in case something happens to the copy you have in the design. Also, you can use the ASCII copy to use as a board for other designs.

1. Choose the **File > Save > Save ASCII Geometries...** menu item. In the dialog box, enter the following, then **OK** the dialog box.

Geometries to Save: **Specific Geometries**
Geometry Names: **signal_analyzer**
Library to Store the Geometry: **User**
Directory in Library: **mgc.trng**
Replace Existing File(s)

A Report-Message window appears confirming that the file was written to the desired location. Because the only new geometry is the board, there is no need to resave all the other geometry in the ASCII libraries. Previously when you saved your geometry to your `trng` library directory, you specified the library name as *Other*, and then you specified the entire pathname to the `trng` directory. This time, because you added the `mgc.trng` User library link, you needed to specify only the name of the library, `User`, and the directory name under `user`, `mgc.trng`.

You entered `LIBRARIAN` on a design, unlike the previous labs. Because the board geometry you just created is for this design, save the geometry to the design as well as to the User library. From now on, when you open the design, the board you see and use is the one you saved with the design, not the one you saved in ASCII format in the `mgc.trng` library. The ASCII format board is the backup. The copy of the board you save with the design is in binary format.

Now you are ready to save the board into the design. Because the board requires other geometries (conn_DIN96, th055028, and so on), they must also be open and checked and saved with the board in the design. You checked all the necessary geometries in the previous procedure.

2. Choose the **File > Save > Design All** menu item.

Whenever you invoke LIBRARIAN on this design, LIBRARIAN automatically reads the board and its related geometries from the binary design objects stored with the design, not from the ASCII parts libraries.

The ASCII parts libraries are a source for parts for all designs. The parts you save with a design in binary format are used only by that one design.

3. After you confirm that the geometries are saved both in the ASCII parts libraries and in the design, close the LIBRARIAN session by choosing **Close** from the Window menu button icon (uppermost-left corner icon).

The LIBRARIAN session closes, because you have already saved a *geoms* design object in a previous step. The *geoms* design object is where the parts for the design are saved in binary format. The *geoms* design object is a file within the design directory *sig_az*.

Congratulations! You have completed Lab 4: "Creating Board Geometries". Continue with Lesson 5: "Mapping and Catalog Files".

Lesson 5

Mapping and Catalog Files

In this lesson, you study mapping and catalog files, which are used by PACKAGE.

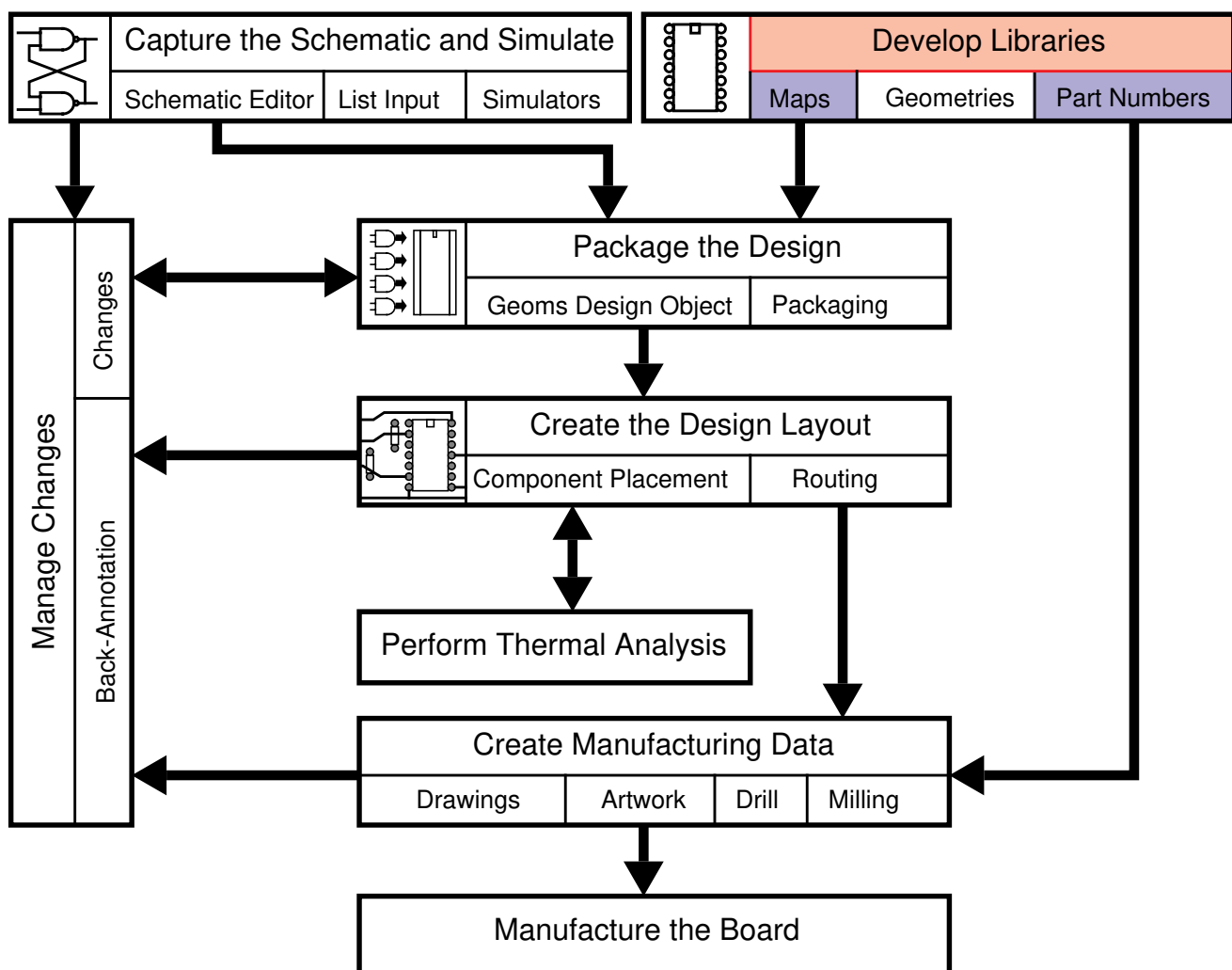


Figure 5-1. PCB Design Process

Objectives

In this lesson, you study the purpose of mapping files and catalog files. You study and practice creating a mapping file and a part number description in a catalog file. You study the default catalog directory hierarchy, where mapping files and catalog files are stored. You also study the advantages of using a design catalog and how to create one.

Upon completion of this lesson and lab, you can:

- Create a catalog directory.
- Create a catalog.
- Add a part number to a catalog file.
- Build the logic description for a schematic symbol.
- Map the symbol to the component geometry.
- Check the catalog and mapping data.

Overview

You create mapping files and catalog files in LIBRARIAN. The PACKAGE tool uses information in a part description when assigning logic symbols from the schematic into geometries to create a components file (*comps* design object). The *comps* design object is input for LAYOUT.

At this point in the design process, you have:

- Logic symbols list for the design.
- Geometries stored in libraries.

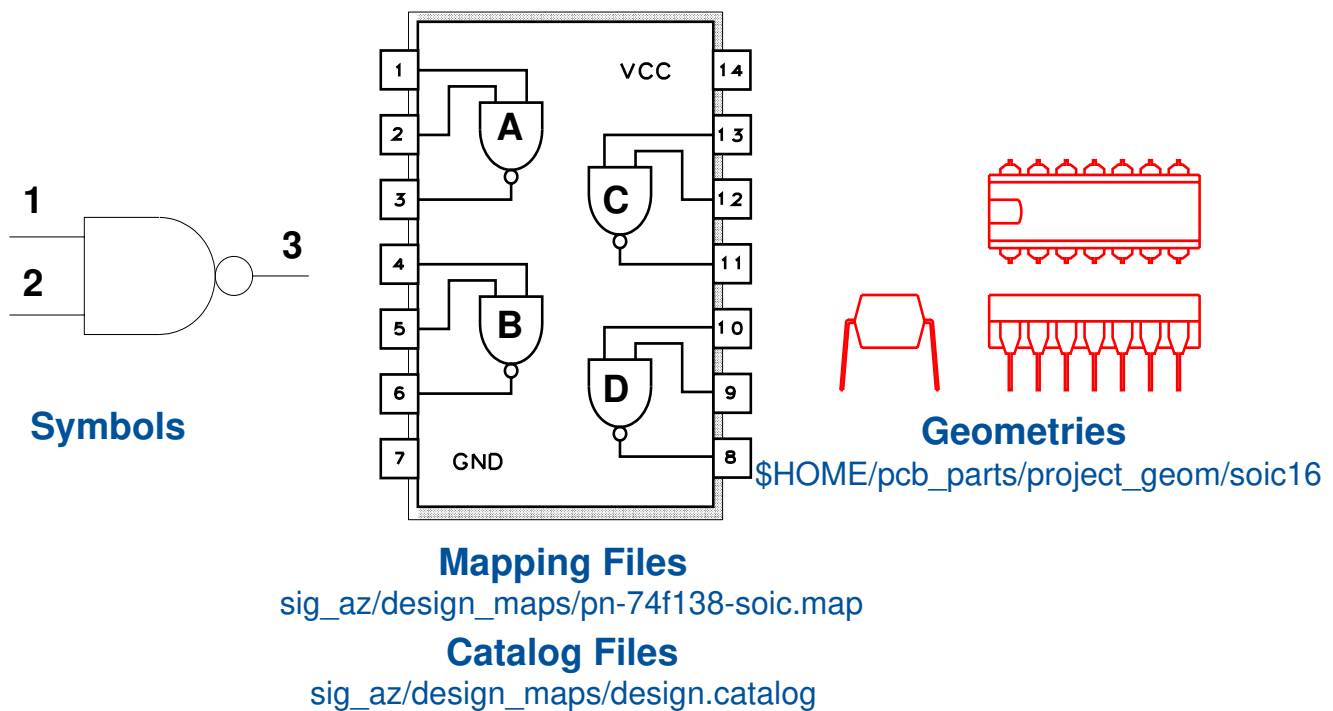


Figure 5-2. Mapping and Catalog Files

Before you can place and route the board, you need a *comps* design object that associates the logic from the symbol instances with the physical footprints represented by the geometries. **Mapping files** and **catalog files** contain the information, or instructions, needed to assign symbol instances to geometries, as shown in Figure 5-2.

Mapping files are ASCII files that state the correlation between the pins of a logic symbol and the pins of a geometry. They also contain swap codes for swapping capabilities in LAYOUT.

Catalog files contain part number descriptions. Each part number description represents a model of an electrical device. The part number description defines the symbol, or symbols, used by that part number, the geometry to which the symbol instances are assigned, the mapping file containing the logical to physical mapping information, and the number of symbol instances to be packaged into a geometry. Other characteristics of the part number (such as the value and rating of resistors) may be present in the description also.

Mapping Files

A mapping file is an ASCII file that states the correlation between the pins of a logic symbol and the pins of a physical component. Each part number in a catalog file references a mapping file. PACKAGE reads the data in the mapping file during the symbol-to-component assignment process, which is called *Build*.

A mapping file does not refer to a specific schematic symbol instance or to a specific component, such as U23. Mapping files are generic. One mapping file can apply to a number of different components and even to different types of logic symbols, provided that the logic symbols have the same pin names and the components have identical pin numbers.

The sample mapping file in Figure 5-3 maps the pins from four logic symbols to the pins of a dip14 component geometry. The map symbol names are: symbol 1, symbol 2, symbol 3, and symbol 4. In a mapping file, the map symbol names can be numbers (as shown) or letters (for example, symbol a, symbol b, symbol c, and symbol d).

The number *1* following each symbol name is a symbol swap code. LAYOUT uses the symbol swap code to determine whether one symbol instance (gate) can be exchanged with another symbol instance to reduce routing congestion in the design. A symbol swap code can be any integer from 0-32767, or the swap code can be blank. A swap code of zero or a blank swap code indicates that swapping is not allowed. Eligible symbols can swap if the symbols are identical, the symbols are packaged in components with the same part number, and the symbol swap codes are the same.

```
#mapping file for 74ls00
```

```
#
```

```
symbol 1 1
```

```
pin "out" "3" 0
```

```
pin "i1" "2" 1
```

```
pin "i0" "1" 1
```

```
symbol 2 1
```

```
pin "out" "6" 0
```

```
pin "i1" "5" 1
```

```
pin "i0" "4" 1
```

```
symbol 3 1
```

```
pin "out" "8" 0
```

```
pin "i1" "10" 1
```

```
pin "i0" "9" 1
```

```
symbol 4 1
```

```
pin "out" "11" 0
```

```
pin "i1" "13" 1
```

```
pin "i0" "12" 1
```

```
power "ground" "7"
```

```
power "vcc" "14"
```

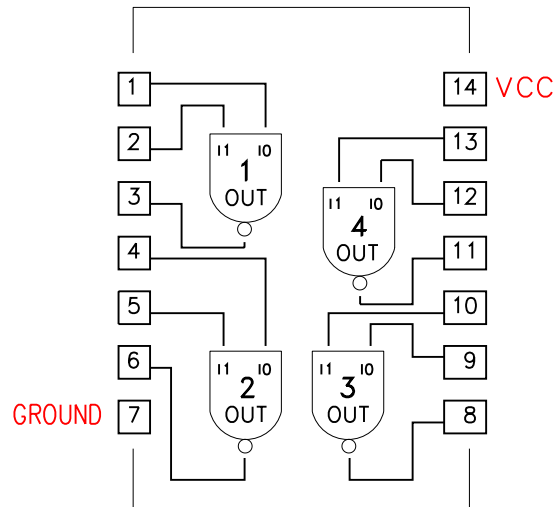


Figure 5-3. Example Mapping File for 74ls00

Syntax: pin “pin_name” “pin_number” “swap_code”

The lines after a map symbol name list the symbol's pins, as shown in Figure 5-4. The sample mapping file shows the pins of symbol 1 as:

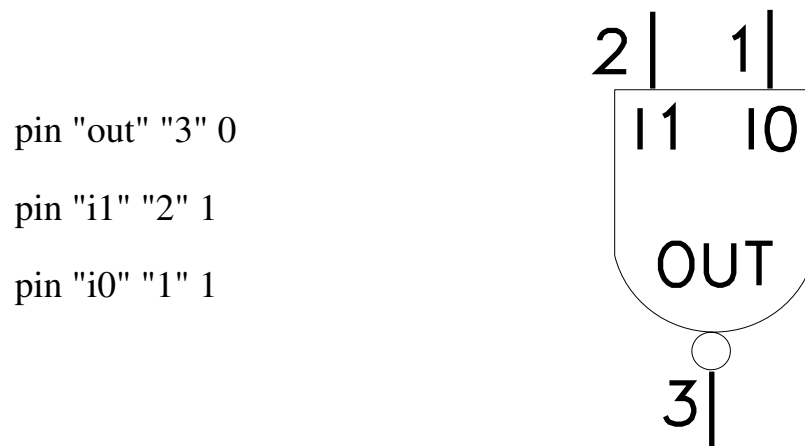


Figure 5-4. Pin Mapping for Symbol 1

Symbol 1 has three pins named *out*, *i1*, and *i0*. The number following the pin name represents a pin number on the dip14 component and shows the association of the logic pin to a physical pin. In this file:

logic pin *out* maps to physical pin 3
 logic pin *i1* maps to physical pin 2
 logic pin *i0* maps to physical pin 1

This format repeats for the logic pins of the remaining symbols.

The number at the end of each pin line is a pin swap code. LAYOUT uses the pin swap code to determine whether one pin can be exchanged with another pin to reduce routing congestion in the design. A pin swap code can be any integer from 0-32767, or the swap code can be blank. A swap code of zero or a blank swap code indicates that swapping is not allowed. Eligible pins with the same swap code can swap within a symbol.

The last two lines of the sample file designate the mapping of the power pins for the component package. The format of the power statement is:

power "power_net_name" "physical_pin_number"

For example:

power "ground" "7"

Comp Property

Board Station uses an identifier called the Comp property to identify symbols. On Mentor Graphics–supplied symbols, the Comp property exists on the symbol. The value of the Comp property is usually similar to the symbol name. For example, the Comp property value of the 74ls00 symbol is 74ls00.

When getting its symbol information from a netlist, rather than from a Mentor Graphics-generated schematic, Board Station uses the symbol name as this identifier. The symbol name is used as the Comp property value would be used from a Mentor Graphics–supplied symbol.

The Comp property value is used in both mapping and catalog files. When creating a mapping file, the Comp property value is used, as an alternative to the symbol name, to identify the symbol containing the logical pin information to map to the physical pin information of the geometry. In catalog files, the Comp property value is used to reference the symbol, or symbols, used for a particular part number.

Non-homogeneous Mapping File

In the previous mapping file, the four symbols are the same. A mapping file that includes a mix of symbols or versions of a symbol is called a non-homogeneous mapping file.

The example part shown in Figure 5-5 contains symbols *inv*, *and*, and *nand4*. You can see in the example that the symbol statement lines include a value for the symbol's Comp property. The Comp property value follows the symbol swap code.

```
# example mapping file
# for non-homogeneous
# part
symbol A 1 "inv"
pin "in" "1" 0
pin "out" "2" 0
symbol B 1 "inv"
pin "in" "3" 0
pin "out" "4" 0
symbol C 0 "and"
pin "i0" "5" 1
pin "i1" "6" 1
pin "out" "8" 0
symbol D 0 "nand4"
pin "i0" "13" 1
pin "i1" "12" 1
pin "i2" "11" 1
pin "i3" "10" 1
pin "out" "9" 0
power "ground" "7"
power "vcc" "14"
```

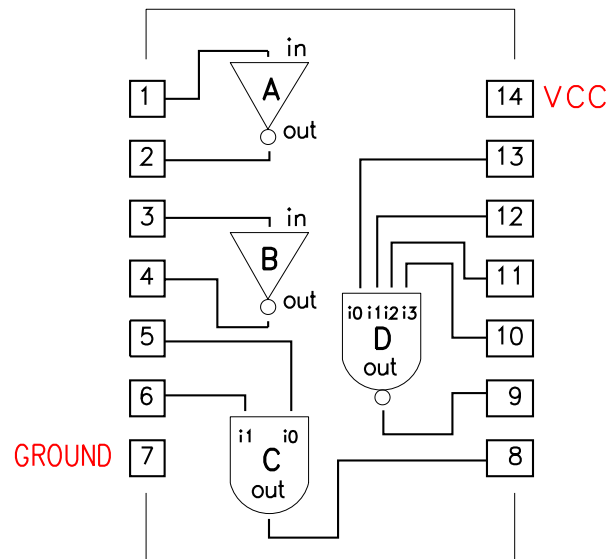


Figure 5-5. Non-Homogeneous Mapping File

Common Pins

Ordinarily, a one-to-one relationship exists between symbol pins and component pins. You can, however, tie a number of symbol pins to the same component pin. These are referred to as common pins. You indicate that a mapping file contains common pins by starting the mapping file with a Common statement using the following syntax:

```
common logical_pin_name physical_pin_number
```

In the example in Figure 5-6, four resistors are packaged into one component. The logical pin names on the resistors are *A* and *B*. All of the *A* pins are tied together and connected to physical pin 1.

```
common A 1
```

```
symbol 1 1
pin "A" "1" 0
pin "B" "2" 0
```

```
symbol 2 1
pin "A" "1" 0
pin "B" "3" 0
```

```
symbol 3 1
pin "A" "1" 0
pin "B" "4" 0
```

```
symbol 4 1
pin "A" "1" 0
pin "B" "5" 0
```

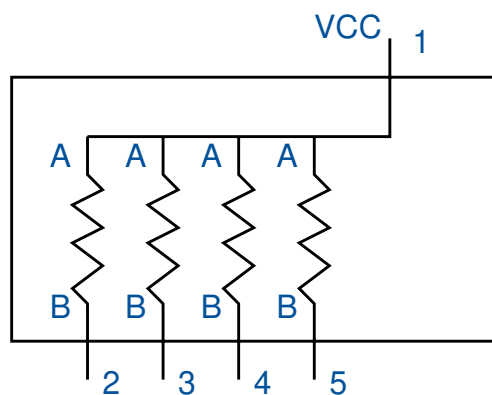


Figure 5-6. Mapping File Using Common Pins

The first line of the mapping file is: `common A 1`

This indicates that the logical *A* pins of the symbols connect to physical pin 1 of the component. You can see under each symbol portion in the mapping file that *pin A* maps to pin 1. Without the common statement, an error would result from more than one logic pin mapping to the same physical pin.

Pinsets

Defining pinsets is a way of associating functional groups of pins within a symbol. The purpose of pinsets is to allow pins to swap as a group.

A pinset definition line in a mapping file consists of the pinset statement, followed by the name of the pinset, a swap code, the pin names of the pins in the pinset, and optionally, a control pin name (refer to Figure 5-7).

Syntax:

```
pinset pinset_name swap_code pin_names... [control_pin_name]
```

```
# MAP FILE: 74ls244a.map
```

```
#
```

```
symbol A 0 "74ls244a"
```

```
pin "1A4" "8" 0
```

```
pin "1A3" "6" 0
```

```
pin "1A2" "4" 0
```

```
pin "1A1" "2" 0
```

```
pin "_1G" "1" 0
```

```
pin "1Y4" "12" 0
```

```
pin "1Y3" "14" 0
```

```
pin "1Y2" "16" 0
```

```
pin "1Y1" "18" 0
```

```
pin "2A1" "11" 0
```

```
pin "2A2" "13" 0
```

```
pin "2A3" "15" 0
```

```
pin "2A4" "17" 0
```

```
pin "2Y4" "3" 0
```

```
pin "2Y3" "5" 0
```

```
pin "2Y2" "7" 0
```

```
pin "2Y1" "9" 0
```

```
pin "_2G" "19" 0
```

```
pinset "ps1" 1 "1Y1" "1A1" ["_1G"]
```

```
pinset "ps2" 1 "1Y2" "1A2" ["_1G"]
```

```
pinset "ps3" 1 "1Y3" "1A3" ["_1G"]
```

```
pinset "ps4" 1 "1Y4" "1A4" ["_1G"]
```

```
pinset "ps5" 1 "2Y1" "2A1" ["_2G"]
```

```
pinset "ps6" 1 "2Y2" "2A2" ["_2G"]
```

```
pinset "ps7" 1 "2Y3" "2A3" ["_2G"]
```

```
pinset "ps8" 1 "2Y4" "2A4" ["_2G"]
```

```
pinset "ps9" 2 "_1G" "1Y1" "1A1" "1Y2" "1A2" "1Y3" "1A3" "1Y4" "1A4"
```

```
pinset "ps10" 2 "_2G" "2Y1" "2A1" "2Y2" "2A2" "2Y3" "2A3" "2Y4" "2A4"
```

```
power "vcc" "20"
```

```
power "ground" "10"
```

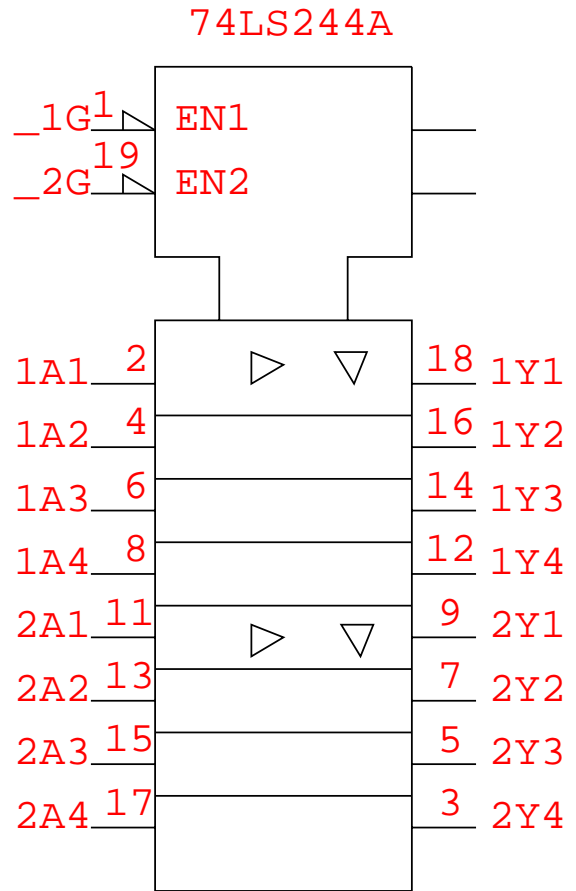


Figure 5-7. Mapping File with Pinsets

Mapping file 74ls244a.map, in Figure 5-7, contains ten pinset definitions. Each 1A series pin is paired with its corresponding 1Y series pin. For example, pins 1A1 and 1Y1 are defined as pinset ps1; pins 1A2 and 1Y2 are defined as pinset ps2. Each 1A/1Y pinset has _1G as the control pin. Pinsets ps1, ps2, ps3, ps4, ps5, ps6, ps7 and ps8 all have swap codes of 1. The same swap code allows swapping between pinsets ps1 through ps8.

Like the 1A series, each 2A series pin is paired with its corresponding 2Y series pin. For example, pins 2A1 and 2Y1 are defined as pinset ps5; pins 2A2 and 2Y2 are defined as pinset ps6. Each 2A/2Y pinset has _2G as the control pin.

The 1A/1Y series pinsets and the 2A/2Y series pinsets all have the same swap code. The LAYOUT tool checks the net. If control pin _1G and control pin _2G are connected to the same net, LAYOUT might swap a 1 series pinset with a 2 series pinset, provided the pinsets have the same swap code. If the control pins are not on the same net, LAYOUT does not swap a 1 series pinset with a 2 series pinset, even if they have the same swap code.

In addition to the pinset pairs, the entire 1 series plus its control pin is defined as pinset ps9, and the entire 2 series plus its control pin is defined as pinset ps10. Both pinset ps9 and pinset ps10 are assigned a swap code of 2. LAYOUT might swap the entire 1 series, defined as ps9, with the entire 2 series, defined as ps10. Note that for pinsets ps9 and ps10, the control pins are treated as ordinary logic pins.

The mapping file 74ls244.map shown in Figure 5-8 describes the association of the logic and physical pins for another version of the 74ls244 symbol. This symbol contains four line drivers with a common enable line. Two symbol functions are packaged in a component.

```

# MAP FILE: 74ls244.map
#
symbol A 1 "74ls244"
  pin "_G" "1" 0
  pin "A1" "2" 0
  pin "Y1" "18" 0
  pin "A2" "4" 0
  pin "Y2" "16" 0
  pin "A3" "6" 0
  pin "Y3" "14" 0
  pin "Y4" "8" 0
  pin "A4" "12" 0
  pinset "s1" 1 "A1" "Y1" ["_G"]
  pinset "s2" 1 "A2" "Y2" ["_G"]
  pinset "s3" 1 "A3" "Y3" ["_G"]
  pinset "s4" 1 "A4" "Y4" ["_G"]
symbol B 1 "74ls244"
  pin "_G" "19" 0
  pin "A1" "11" 0
  pin "Y1" "9" 0
  pin "A2" "13" 0
  pin "Y2" "7" 0
  pin "A3" "15" 0
  pin "Y3" "5" 0
  pin "Y4" "3" 0
  pin "A4" "17" 0
  pinset "s1" 1 "A1" "Y1" ["_G"]
  pinset "s2" 1 "A2" "Y2" ["_G"]
  pinset "s3" 1 "A3" "Y3" ["_G"]
  pinset "s4" 1 "A4" "Y4" ["_G"]
power "ground" "10"
power "vcc" "20"

```

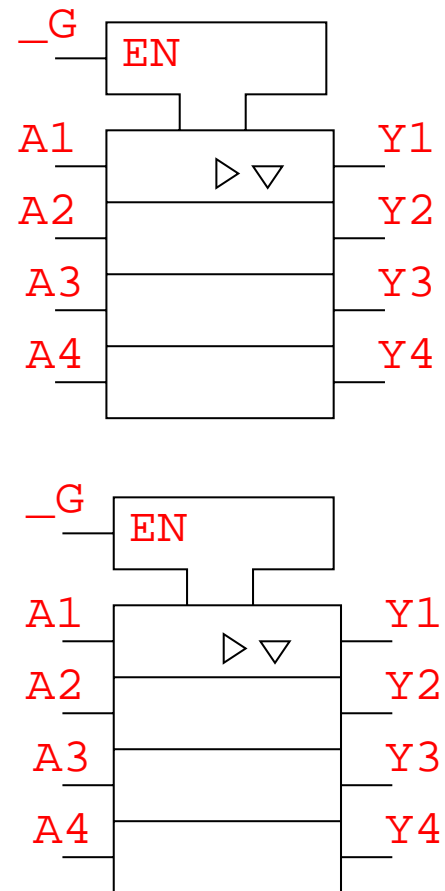


Figure 5-8. Mapping File with Pinsets for Each Symbol

The pinset definitions control the swapping within each symbol. A pinset is defined for each of the four line driver functions in the symbol. For example, pinset s1 is defined as logic pins A1 and Y1 and control pin _G. Using the same swap code for each pinset allows each pinset to swap within the symbol.

Figure 5-9 shows another mapping for the 74ls244 component. This mapping file does not show an example of pinsets, but the mapping file does show the remaining form of a 74ls244 symbol that is typically found on schematics.

In this example, each line driver is represented by a separate symbol group. The swap codes for symbols A through D are the same. These four symbols are allowed to swap. The swap codes for symbols E through H are the same, but they differ from the first set of four. The symbols E through H swap independently of the symbols A through D.

```

# MAP FILE: 74ls244b.map
common "_G" "19"
common "_G" "1"
symbol A 1 "74ls244b"
  pin "A" "2" 0
  pin "Y" "18" 0
  pin "_G" "1" 0
symbol B 1 "74ls244b"
  pin "A" "4" 0
  pin "Y" "16" 0
  pin "_G" "1" 0
symbol C 1 "74ls244b"
  pin "A" "6" 0
  pin "Y" "14" 0
  pin "_G" "1" 0
symbol D 1 "74ls244b"
  pin "A" "8" 0
  pin "Y" "12" 0
  pin "_G" "1" 0
symbol E 2 "74ls244b"
  pin "A" "11" 0
  pin "Y" "9" 0
  pin "_G" "19" 0
symbol F 2 "74ls244b"
  pin "A" "13" 0
  pin "Y" "7" 0
  pin "_G" "19" 0
symbol G 2 "74ls244b"
  pin "A" "15" 0
  pin "Y" "5" 0
  pin "_G" "19" 0
symbol H 2 "74ls244b"
  pin "A" "17" 0
  pin "Y" "3" 0
  pin "_G" "19" 0
power "ground" "20"
power "vcc" "10"

```

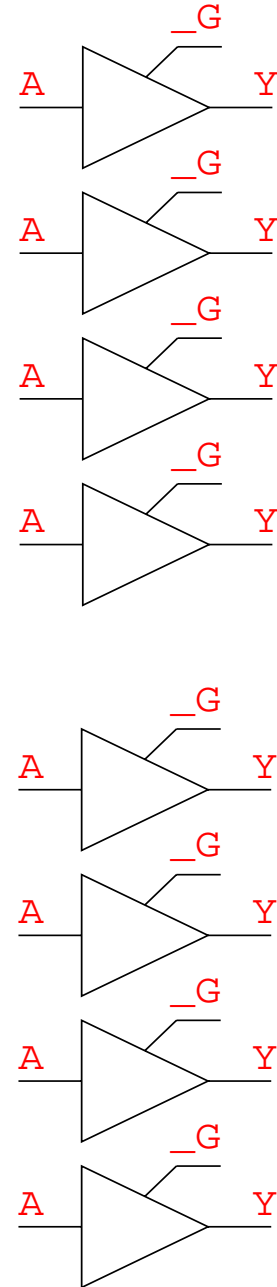


Figure 5-9. 74ls244 Mapping File without Pinsets

Catalog Files

A catalog file is a collection of part numbers. An example is shown in Table 5-1. Each part number represents a model of an electrical device (component). The part number description consists of fields of information that identify the logic symbol (Comp property), component geometry, the mapping file containing the logical to physical mapping information for this component model, and the symbol count. In the PCB design process, catalog file data is input to PACKAGE, where the part numbers are translated to specific component devices in your design. The part description identifies:

- Type of symbol or symbols to use based on the Comp property value or values.
- Geometry of the component.
- Mapping file to follow for the symbol pin to component pin assignments.
- Number of symbol instances required.
- Any properties assigned to the part.

The catalog file is used by PACKAGE to assign symbol instances from the schematic to components that will be placed and routed on the board. PACKAGE finds the correct part number to use by looking at the Comp property value of a symbol and sorting through the loaded catalog files to find a part description that calls for a symbol with that Comp property value. You can refine the selection of a part number by adding other properties to the symbol as additional criteria in the selection process.

Table 5-1. An Example Catalog File

```
#!/ PART NUMBER CATALOG FOR DESIGN.CATALOG
```

```
#
```

PART_NO	COMP_PROPERTY	GEOMETRY	MAPPING_FILE	SYMB_CNT
pn-7408	7408	dip14	7408.map	4
pn-7432	7432	dip14	7432.map	4
pn-7474	7474	dip14	7474.map	2
pn-74ls00	74ls00	dip14	74ls00.map	4
pn-74ls151	74ls151	dip16_p	74ls151.map	1
pn-capacitor	capacitor	cs13a	cs13a.map	1
pn-capacitor_a	capacitor_a	ck50	ck50.map	1
pn-conn2X32	edge	conn2x32	edge_conn.map	64
pn-pullup	pullup	rc07	rc07.map	1

Each line in the catalog file is a part number. The fields of information in the part numbers are:

- **PART_NO**—the identification for the description.

The part number is a unique identifier for each part description in the catalog file. You can use any characters in the part number, but characters other than A-Z, a-z, 0-9, \$, and _ are considered special characters and must be enclosed in quotes. A dash (-) can be used, provided that the dash is not the first character of the part number. The part numbering convention of the Mentor Graphics-supplied part descriptions is Pn-Comp_property_value. For example, Pn-74ls00.

You can create multiple versions of the same part. For example, you might have a resistor that is usually placed horizontally, but the resistor can be placed vertically if space on the PC board is limited. For each case, the part description calls for a different geometry.

You create versions of part numbers by appending the pipe character (|) and additional text to the part number. For example,

Pn-res|h

Pn-res|v

The PACKAGE tool writes the versioned part numbers to the *comps* design object as the same part number by truncating the '`<text>`'.

If your standard part numbering convention includes the pipe (|) character, add an extra | character at the end of the part numbers to prevent PACKAGE from truncating characters that are required for recognition of the part. For example, a part number that is normally Pn-7408|09 must be specified as Pn-7408|09|.

- **COMP_PROPERTY**—the name of the logic symbol or the Comp property value attached to the logic symbol that represents the electrical functions of the component.

The PACKAGE tool uses the Comp property (or equivalent) of a symbol to select a part description from the catalog file.

In the part number description for a non-homogeneous part number, the Comp_property field contains the Comp property values for all the symbols in the part. The values are enclosed in parentheses and separated by a comma, space, or both. For example, (74ls14, 74ls09, 74ls13).

- **GEOMETRY**—the name of the component geometry that represents the physical package of the component.

The geometry also represents the component outline and pin locations, such as dip14 or switch. A geometry does not have to exist at the time the catalog file is created; you can create the geometries later, but all geometries must exist before your design enters LAYOUT.

- **MAPPING_FILE**—the name of the mapping file that assigns the logic pins to the physical pins.

The mapping file defines the symbol pin to component pin association for this component part. All mapping files named in a catalog file must reside in the same catalog directory as the catalog file.

If a component has many parts (ASIC, gate arrays, and so on) or for parts whose pin numbers are changing on the schematic, you might not want an ASCII mapping file. To have Build get pin numbers directly from the schematic instead of from a mapping file, enter the value *nomap* for the name of the mapping file.

Table 5-2. No Mapping File

```
# PART NUMBER CATALOG FOR DESIGN.CATALOG
#
```

PART_NO	COMP_PROPERTY	GEOMETRY	MAPPING_FILE	SYMB_CNT
pn-74hct138	74as138	soic16	nomap	1

When using *nomap* to get pin information:

- Pin numbers must exist on the schematic.
- You cannot use wildcards for pin numbers on schematic.
- You must either route a pin to power, or add the *power_pin* property to a pin on the schematic.
- No swapping is allowed.
- No checking occurs.
- Build assumes only one symbol exists per package.
- **SYMB_CNT**—the number of symbol instances in the component.

You can add optional information to a part number for property data and comments. Property fields have no headings.

Catalog Properties

Property names and values appear either to the right of the Symb_cnt field, or below the part number in the format:

property_name = property_value

During the automatic assignment process (Build), PACKAGE attempts to determine the part number of all unspecified gates, such as resistors, by matching symbol property data against corresponding properties in the catalog file or files and matching the geometry, if it is available. The search is based on the first catalog entry with properties that are the same as or a superset of the gate's properties. The search ends on the first valid match.

If you enter property data, make sure the catalog properties are as complete as possible and that *default* catalog part numbers come first in a catalog. The default condition for a property search is to select the first part number that matches a superset of the symbol's properties. For example, a RES symbol with the property Value = 1k would match an entry in the catalog file that has the properties *Value = 1k* and *Rating = 1/2W*.

User Comments in Catalog Files

You can add comments to the catalog file before the header line at the beginning of the file or after any part number to which the comment applies. Use the MGC Notepad to add and edit comments in a catalog file. Comments must begin with the characters #!.

Catalog Libraries

Mapping files and catalog files are stored in directories, like geometries. A catalog directory contains a catalog file and one or more mapping files associated with that catalog file. You can assign any legal filename to a catalog file, but the filename must include the *.catalog* extension. A mapping file can have any legal filename. By default, LIBRARIAN names mapping files using the convention *<symbol_name>.map*.

Figure 5-10 shows the structure and naming conventions for a catalog directory and its mapping files and catalog file. The catalog, or catalog directory, is named *<name>_maps*. The mapping files within the catalog are named *<anyname>.map*. The catalog file has the same name as the catalog directory, but with a *.catalog* extension, *<name>.catalog*.

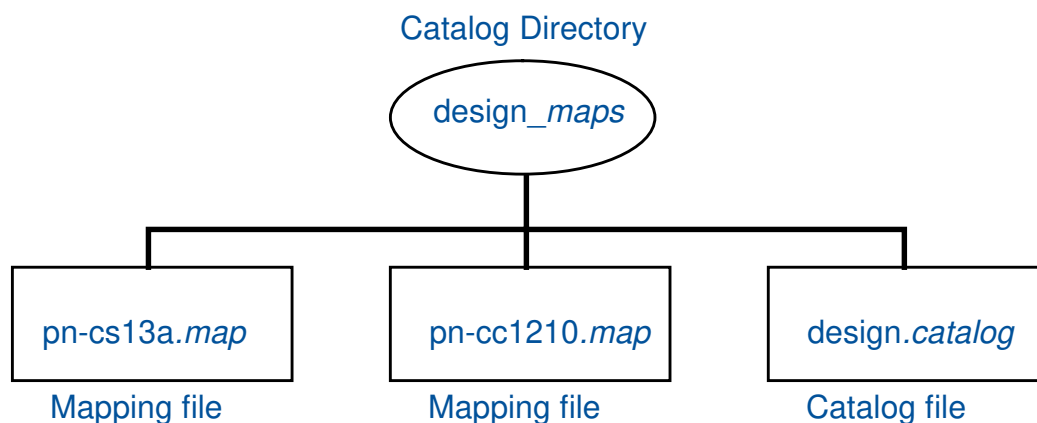


Figure 5-10. A Catalog Directory Structure

Catalog files can be located anywhere in your system network. All of the mapping files associated with a given catalog file must be located in the same catalog directory as that catalog file. This is different from the rest of the files for a given design, all of which must be located under a single system directory whose name is the design name.

Default Directory Hierarchy

Catalog and mapping files are found at each level of the hierarchy in the directory named:

<level_name>_maps

Refer to Table 5-3. The name of the catalog directory is slightly different at the Mentor level of the hierarchy. At the Mentor level, the <level_name> becomes pcb, as follows:

<level_name>_maps is *pcb_maps*

Mapping and catalog files offered by Mentor Graphics are located in the directories under the *\$MGC_PCBPARTS/pcb_maps* directory.

Table 5-3. An Example of the Default Directory Hierarchy

Directory Level	Default Pathname
Mentor	\$MGC_PCBPARTS/pcb_geoms
	\$MGC_PCBPARTS/pcb_maps
	\$MGC_PCBPARTS/pcb_libs
Company (optional)	\$HOME/pcb_parts/company_geom
	\$HOME/pcb_parts/company_maps
	\$HOME/pcb_parts/company_lib
Project (optional)	\$HOME/pcb_parts/project_geom
	\$HOME/pcb_parts/project_maps
	\$HOME/pcb_parts/project_lib
User	\$HOME/pcb_parts/user_geom
	\$HOME/pcb_parts/user_maps
	\$HOME/pcb_parts/user_lib

Table 5-3. An Example of the Default Directory Hierarchy

Directory Level	Default Pathname
Design	design_pathname/ <i>design_geom</i>
	design_pathname/ <i>design_maps</i>
	design_pathname/ <i>design_lib</i>
Other	any_pathname

Depending on how your system manager sets up your system, these directories (and links) are automatically created when you invoke LIBRARIAN. In the *\$HOME/pcb_parts* directory, LIBRARIAN creates the *user_maps* directory. LIBRARIAN also creates the directory *design_maps* in your design directory, if the directory does not exist when you invoke LIBRARIAN on a design. Optionally, LIBRARIAN creates directories or links to directories at the Company and Project levels.

The directory level called Other is for users who do not want to use the default directory hierarchy, or who have existing directories and files outside of the default directory hierarchy. The Other directory does not have a predefined location, rather it is simply a category that allows you to enter an absolute pathname to access data outside the directory hierarchy. LIBRARIAN stores the pathnames to catalogs accessed at the Other directory level (if the pathnames have been read into the LIBRARIAN session) and restores the pathnames at the next LIBRARIAN session. LIBRARIAN writes the pathnames in the file *\$HOME/pcb_parts/other_pathnames*.

Using the Default Directory Hierarchy

Using the default directory hierarchy, you can read and write files in the libraries without knowing the absolute pathnames to the files. You can view the default catalog directory hierarchy at any time by selecting menu item:

Catalogs > List Catalogs...

A dialog box opens, as shown in Figure 5-11, to show the levels in the default directory hierarchy.

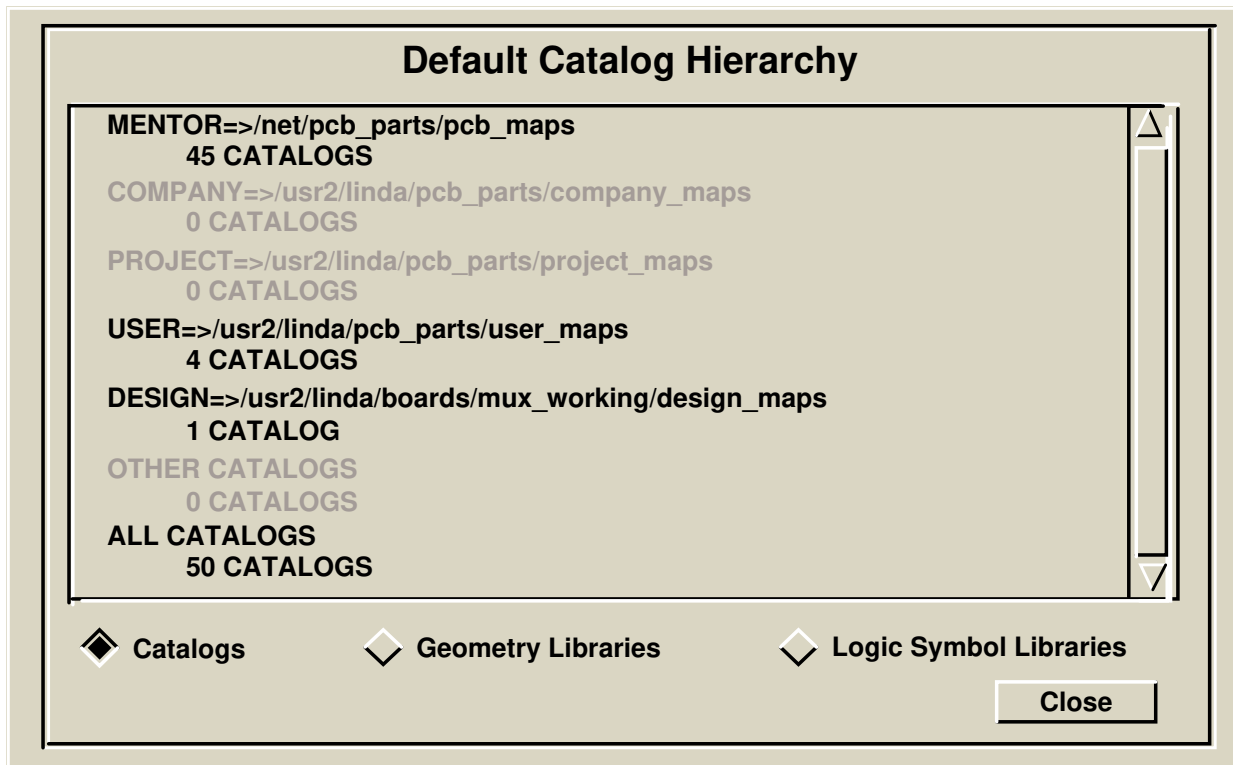


Figure 5-11. Default Catalog Hierarchy Dialog Box

The dialog box dims the level name of any level that has no catalog data. The levels that currently contain catalog data are available for selection. Available levels can differ from session to session, and from user to user, because the levels represent directories that can be modified (except the Mentor level). Along with the default directory

level names, the dialog box also displays the pathnames for each level and reports the number of catalogs at each level.

To see a listing of the catalogs at one of the levels, select the level by placing the cursor over the level name and clicking the Select mouse button.

The display in the dialog box changes to show a listing of the catalogs at the selected level, as shown in Figure 5-12. This listing of catalogs indicates whether a catalog has been read into the LIBRARIAN session. The listing also shows whether one of the catalogs is the active catalog.

You can select any or all of the catalogs in the listing. Pressing the Read button at the bottom of the dialog box reads a selected catalog into the LIBRARIAN session. Pressing the View button lists the contents of a selected catalog.

From the listing of the catalog contents, you can select and read one or more items in the listing.

To exit the dialog box, you press the Close button at the bottom of the dialog box.

Concept of Active

The concept of *active* is important to catalogs and part numbers. At any time, you can have an active catalog, an active part number, or both.

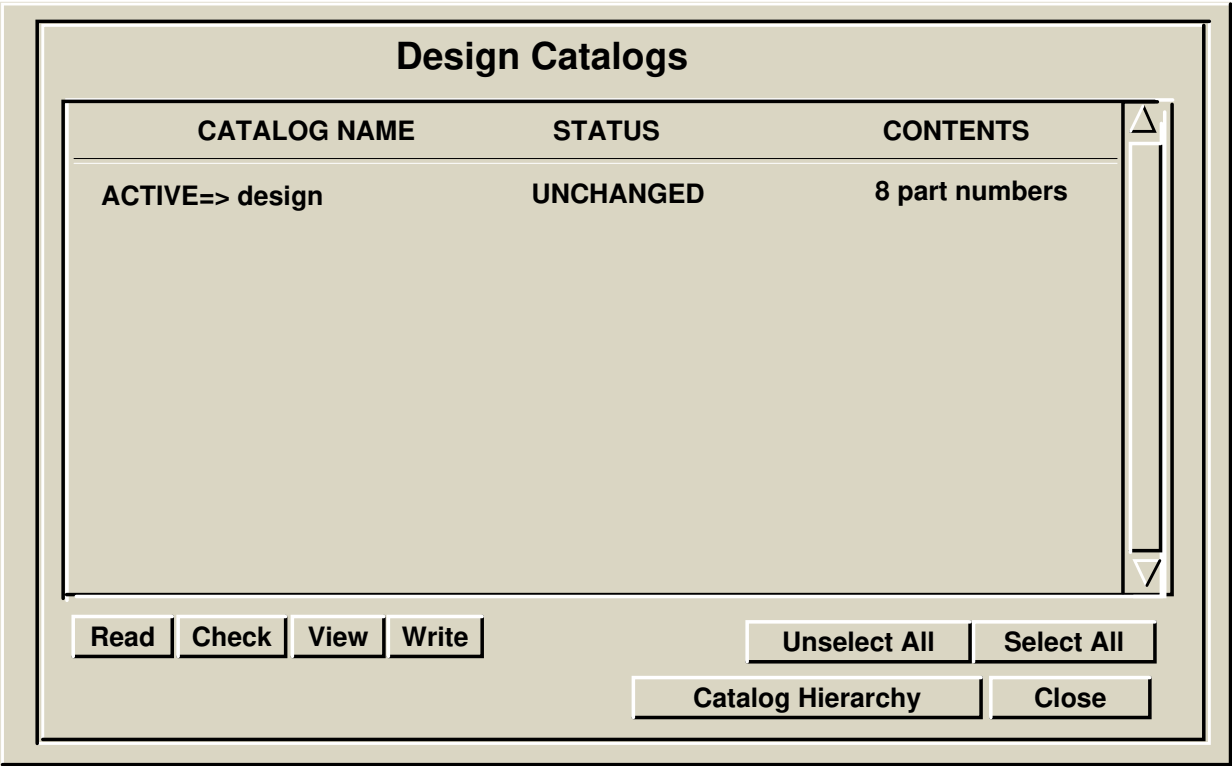


Figure 5-12. Listing of Catalogs with the Active Catalog Indicated

A catalog becomes active when you read or view the library or catalog and when you save data to the catalog. A catalog remains active until you read, view, or save data to another catalog. The data in a previously active catalog remains available for use in the LIBRARIAN session.

List Active Catalog

LIBRARIAN tracks the active data and provides direct access to an active catalog with the menu item:

Catalogs > List Active Catalog...

When you list an active catalog, you see a dialog box similar to that shown in Figure 5-13. When a catalog is active, you can create a part number in the catalog. A part number that you read or create becomes the active file or part number. The part number remains active until you read or create another part number. If you have multiple part numbers open, the part number in the active window becomes the active part number.

Several of the menu selections for checking and reporting provide an option to focus on the active data.

Part Numbers List				
PART NUMBER	STATUS	GEOMETRY	COMP PROPERTY	MAP FILE
DESIGN CATALOG design				
pn-7408	UNREAD	dip14	7408	7408.map
pn-7432	UNREAD	dip14	7432	7432.map
pn-7474	UNREAD	dip14	7474	7474.map
pn-741s151	UNREAD	dip16_p	741s151	741s151.map
pn-capacitor	UNREAD	ck60	capacitor	ck60.map
pn-capacitor-a	UNREAD	cs13_a	capacitor	cs13.map
pn-connx32	UNREAD	connx32	edge	edge_conn.map
pn-pullup	UNREAD	rc07	pullup	rc07.map

CheckDeleteDraw

Unselect AllSelect All

Design CatalogsCatalog HierarchyClose

Figure 5-13. The Contents of the Active Catalog

Creating a Part Number

Creating a part number combines a symbol and a geometry, and creates both a part number description in a catalog file and a mapping file. Five steps must be accomplished to create the part number.

1. Verify the name and directory level of the catalog file in which to store the part number. If no catalog directory exists, you must create one.
2. Read into the session any geometry from the geometry libraries that you are going to use in the part number.
3. Read into the session any symbols that you are going to use in the part number.

Now you are ready to *create* a new part number.

4. Choose the **Catalogs > Create Part Number...** menu item

The Create Part Number dialog box opens as shown in Figure 5-14.

Create Part Number

Part Number

Catalog Name

Geometry

Mapping

Directory

☐ Company

☐ Project

☐ User

☒ Design

☐ Replace Part

Figure 5-14. The Create Part Number Dialog Box

The fields in the dialog box include:

- **Part Number**—you enter the part number here.
 - **Catalog Name**— you enter the name of the catalog file in the entry box. By default, the name of the active catalog file, if any, appears in the entry box.
 - **Directory**—you press the button that identifies the directory level of the catalog file. By default, the button identifying the directory level of the active catalog file is pressed for you.
 - **Geometry**—you enter the name of the geometry for this part number.
 - **Mapping**—you choose one of the following responses for this entry box:
 - Enter the name of an existing mapping file in the entry box.
 - Leave the entry box blank to create a new mapping file.
 - Enter the string, *nomap*.
 - **Replace Part**—press this button to replace an existing part description with the same part number with a new part number.
5. Press the **OK** button to execute the dialog box.

Mapping Windows

After you choose the **Catalogs > Create Part Number...** menu item, and complete the dialog box, two windows open on the screen: an Edit window and a Map window, as shown in Figure 5-15. The Edit window contains the geometry for the part number, if the geometry exists and was read into the session. The Edit window is empty if the geometry does not exist, or if you did not read it into the session.

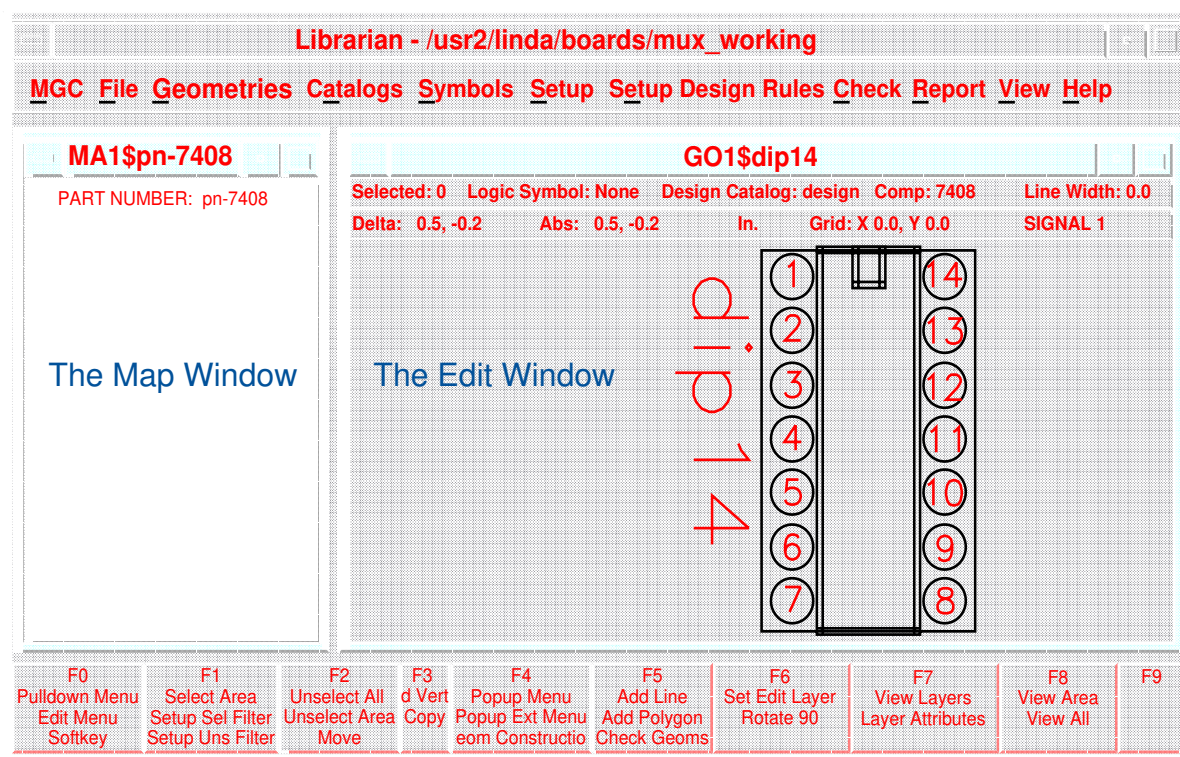


Figure 5-15. The Edit and Map Windows for Creating a Part Number

What you enter in the Mapping entry box of the Create Part Number dialog box determines the contents of the Map window. The Map window display is one of the following:

- An existing mapping file.
- An empty window, except for a line displaying the part number.
- An empty window, except for a line displaying the part number and the designation, <NOMAP>. For more information about *nomap*, refer to list bullet item MAPPING_FILE on page 5-19.

Mapping with a Logic Symbol

Creating a mapping file with a logic symbol means that the logic symbol and pin data in the mapping file are based on symbol data stored in the default logic symbol library hierarchy. You create a mapping file with a logic symbol when the schematic design uses symbols from an available symbol library.

Mapping with a logic symbol is the primary method of mapping logical to physical pins.

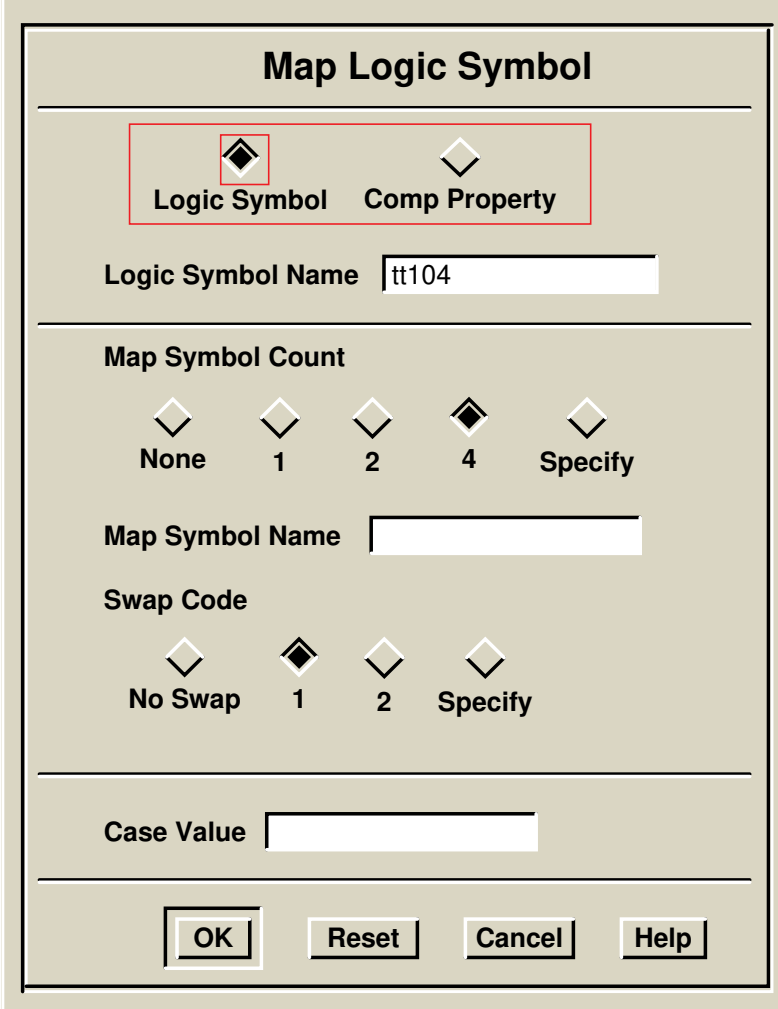
LIBRARIAN automatically reads the logic symbol pin names from symbol data in the library into the mapping file. If LIBRARIAN can determine the logical pin-to-physical pin assignments, LIBRARIAN adds the pin mapping information to the mapping file.

Before mapping the pins of a logic symbol, the following must be complete:

- Read the physical geometry into the session.
- Read the logical symbol into the session.
- Activate a catalog.

With the cursor in the Map window, choose the **Map Logic Symbol** menu item from the popup menu, and fill in the dialog box as follows.

The Map Logic Symbol dialog box opens, as shown in Figure 5-16.



The dialog box is titled "Map Logic Symbol". It contains two radio buttons at the top: "Logic Symbol" (selected) and "Comp Property". Below these is a text field for "Logic Symbol Name" containing "tt104". A horizontal line separates this from the "Map Symbol Count" section, which has five radio buttons: "None", "1", "2", "4", and "Specify". Below this is a text field for "Map Symbol Name". Another horizontal line separates this from the "Swap Code" section, which has four radio buttons: "No Swap", "1", "2", and "Specify". Below this is a text field for "Case Value". At the bottom are four buttons: "OK", "Reset", "Cancel", and "Help".

Figure 5-16. Map Logic Symbol Dialog Box

1. **Logic Symbol**—you identify the logic symbol for the mapping file by its symbol name. Choose **Logic Symbol** and then enter the symbol name.

The symbol name you enter must be a symbol that you have read from the symbol library into the LIBRARIAN session.

2. **Map Symbol Count**—you enter the number of symbol instances (gates) to assign to this part number by selecting *None*, 1, 2, 4, or *Specify*.

If you enter a symbol count other than *None*, the dialog box changes to include the fields that are described in steps c and d.

3. **Map Symbol Name**—you enter the name for the first symbol in the mapping file. If the mapping file has more than one symbol instance, LIBRARIAN automatically increments the subsequent symbol names.

The symbol name can be a letter or letters, a number, or an alphanumeric combination. For example, enter a 1 to name the symbols numerically, or you could enter A to name the symbols alphabetically. Using a name such as J1 names the symbols in the sequence J1, J2, J3,.... Using a name such as AA names the symbols in the sequence AA, AB, AC....

4. **Swap Code**—assign a swap code to the symbol by selecting *No Swap*, 1, 2, or *Specify*.
5. Press the **OK** button to execute the dialog box.

LIBRARIAN adds the logic symbol and logic pin information in the Map window.

Adding Power Pins

You add power pins to a mapping file to assign power nets to physical pins.

1. From the Top Menu (popup menu) in the Map window, choose the **Add Power Pins...** menu item.

The Add Mapping File Power Pins dialog box opens, as shown in Figure 5-17.

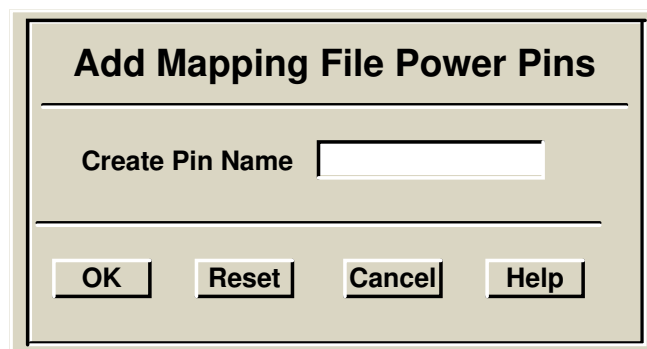


Figure 5-17. The Add Mapping File Power Pins Dialog Box

2. In the **Create Pin Name** entry box, enter the power net name for which to create a pin. For example, vcc or ground.

As you begin typing, the dialog box displays another entry box for entering another power pin name.

3. Press the **OK** button to execute the form and create the entries in the mapping file for the power pins.

The format of the entries is:

```
POWER PIN: <power_net_name> PHY: NONE
```

Mapping Pins and Setting Swap Codes

Pin mapping assigns a logic pin in the mapping file to a physical pin on the component geometry in the Edit window, which is shown in Figure 5-18.

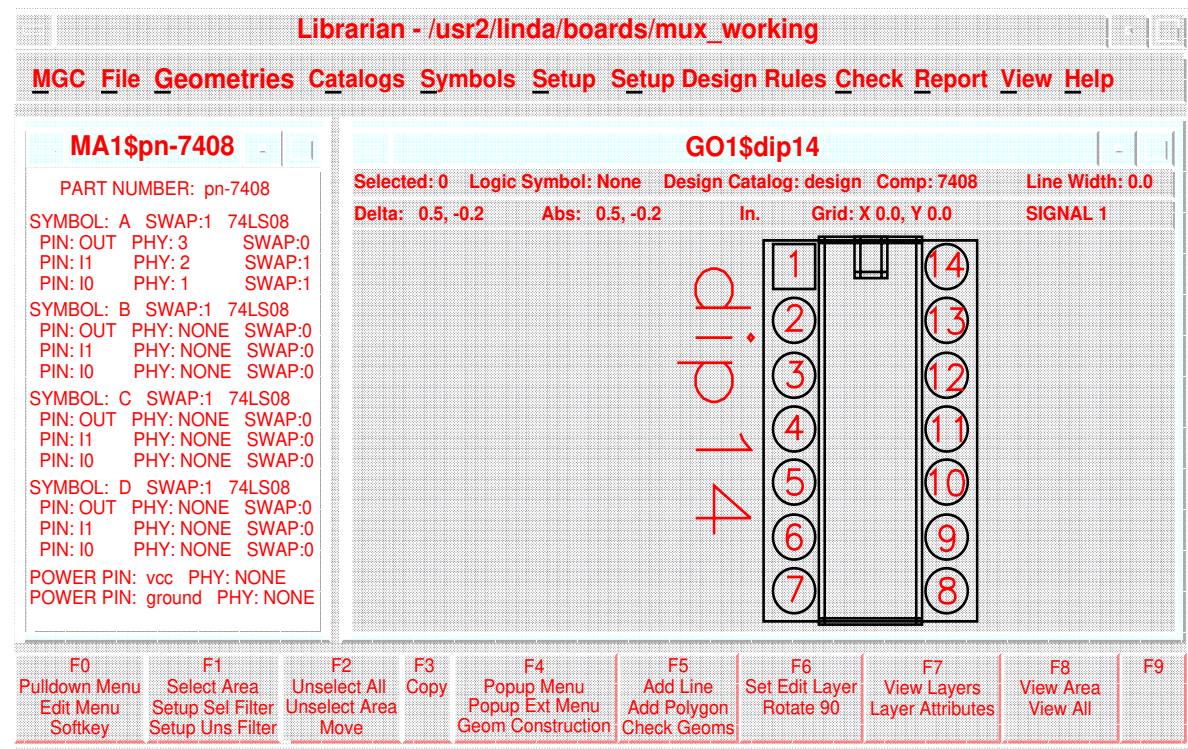


Figure 5-18. The Edit Window used for Pin Mapping

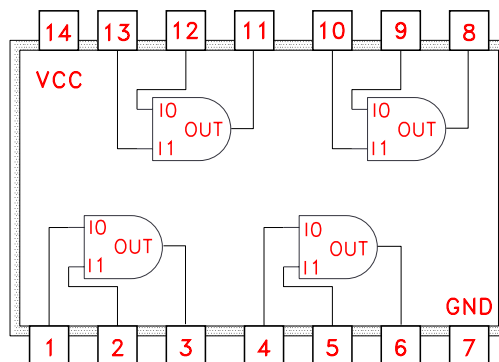


Figure 5-19. The Symbol Connections to be Mapped

1. From the **Top Menu** in the Map window, choose the **Map Pins** menu item.

The Map Pins prompt bar is displayed and prompts for a Logic Pin Location.

2. At the **Logic Pin Location** prompt, move the cross-hair cursor into the Map window and select a logic pin.

Repeat control—You press the stepper buttons to cycle through the options of *repeat* (to map pins in succession) and *norepeat* (to map a single logic pin).

3. The Select Physical Pin prompt bar appears and prompts for the location of a physical pin on the component geometry. Move the cross-hair cursor into the Edit window and select a physical pin.

Setting Pin Swap Codes

The swap code for each pin can be set as follows:

1. From the **Top Menu** in the Map window, choose the **Change Swap Code > For Pins** menu item.

The Select Map Pin prompt bar appears.

2. Move the cross-hair cursor into the Map window and select a logic pin.
3. In the Change Pin Swap Code prompt bar, enter the new swap code.

Completed Mapping Data

All data is added to the mapping file data in LIBRARIAN: the symbols and their swap codes; the pins, the pin numbers, and pin swap codes; and the power statement and pin assignments (refer to Figure 5-20).

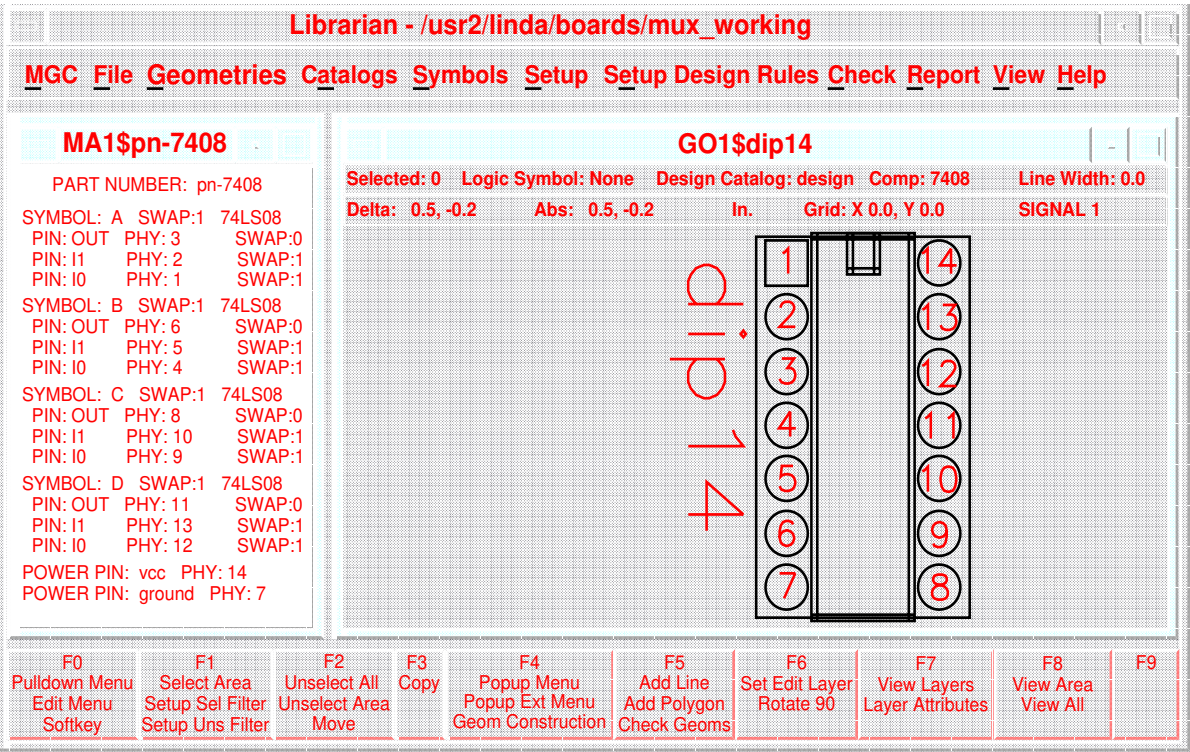


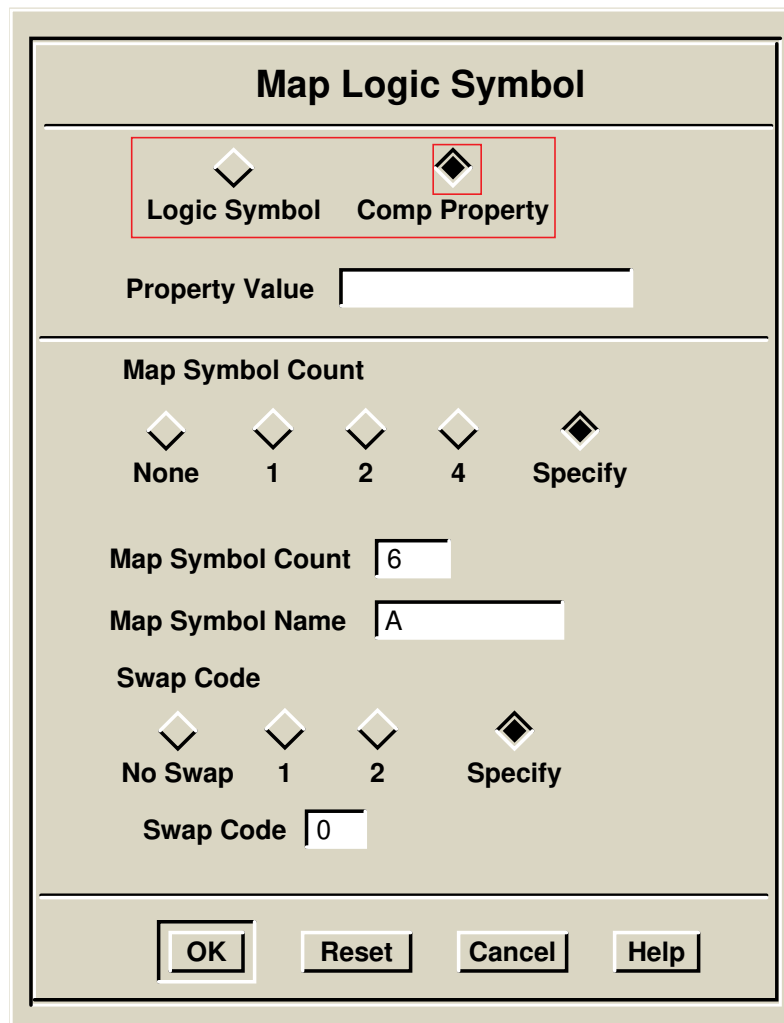
Figure 5-20. The Edit Window with Completed Mapping Data

Mapping Without a Logic Symbol

Creating a mapping file without a logic symbol means that you provide all the logic symbol and pin information for the mapping file. You only use this method for mapping pins when there is no logical symbol data from a design.

To create a mapping file without a logic symbol:

1. From the popup menu in the Map window, choose the **Map Logic Symbol** menu item. The Map Logic Symbol dialog box opens, as shown in Figure 5-21. Complete the dialog box.



The image shows a dialog box titled "Map Logic Symbol". At the top, there are two radio buttons: "Logic Symbol" (unselected) and "Comp Property" (selected). Below these is a "Property Value" text field. The next section is "Map Symbol Count", which includes five radio buttons: "None", "1", "2", "4", and "Specify" (selected). Below this is a "Map Symbol Count" text field containing the number "6". The next section is "Map Symbol Name", with a text field containing the letter "A". The "Swap Code" section has four radio buttons: "No Swap", "1", "2", and "Specify" (selected). Below this is a "Swap Code" text field containing the number "0". At the bottom of the dialog box are four buttons: "OK", "Reset", "Cancel", and "Help".

Figure 5-21. The Map Logic Symbol Dialog Box

- a. In the dialog box, enter either a logic symbol name or a Comp property and value.
 - **Comp Property**—press this button and enter a Comp property value.
 - **Map Symbol Count**—enter the number of symbol instances (gates) to assign to this part number by choosing None, 1, 2, 4, or Specify.

When you enter a symbol count other than *None*, the dialog box changes to include the additional fields:

- b. **Map Symbol Name**—enter the name for the first symbol in the mapping file. If the mapping file has more than one symbol instance, LIBRARIAN automatically increments the subsequent symbol names.
 - c. **Swap Code**—assign a swap code to the symbol by choosing None, 1, 2, or Specify.
2. Press the **OK** button to execute the dialog box. LIBRARIAN updates the mapping file to include map symbol names and swap codes for the logic symbols. Next, add the logic pins to the symbols in the mapping file.
 3. In the popup menu in the Map window, choose the **Add Logic Pins** menu item, and complete the dialog box as follows.

The Add Mapping File Pins dialog box opens, as shown in Figure 5-22.

Add logic pins only when you map without a logic symbol. If you have a logic symbol, and you map from it, the logic pin information comes from the logic symbol directly, so you do not need to add them manually.

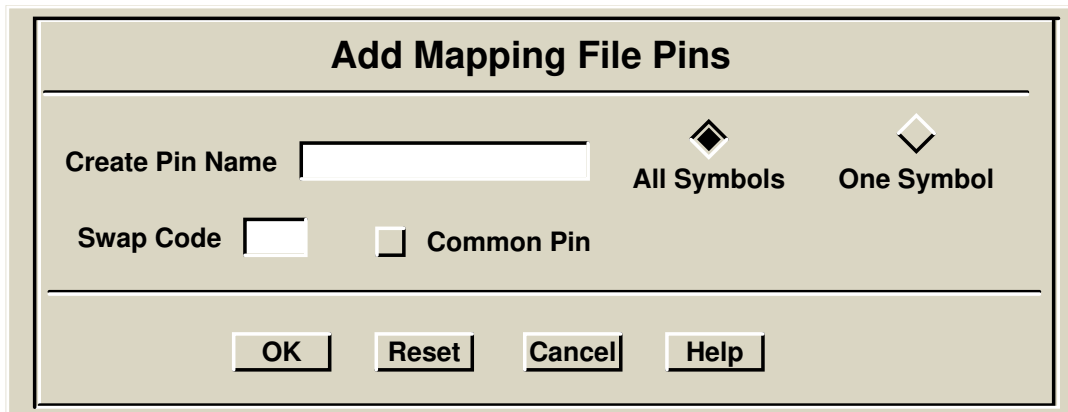


Figure 5-22. The Add Mapping File Pins Dialog Box

- **Create Pin Name**—enter the name of a logic pin. For example, i0 or out.
- **All Symbols**—adds the pin to all the map symbols.
- **One Symbol**—adds the pin to one map symbol in a mapping file with more than one map symbol.

Name—enter the map symbol name of the symbol to receive the pin.

- **Swap Code**—enter a whole number from 0-32767 to assign a swap code to the pin.
- **Common Pin**—press this button if the pin you are creating is a common pin.

COMMON PIN: <PIN_NAME>

The dialog box repeats these controls so you can create additional pins.

4. Press the **OK** button to execute the dialog box. LIBRARIAN adds the logic pins to the mapping file.
5. Add Power Pins and Mapping Pins.

You add power pins to a mapping file to assign power nets to physical pins. Refer to section “Adding Power Pins” on page 5-35. Pin mapping assigns a logic pin in the mapping file to a physical pin on the component geometry in the Edit window. See section “Mapping Pins and Setting Swap Codes” on page 5-36.

Checking Part Number Data

You can check the active part number that you just created. LIBRARIAN checks:

- Logic symbol, if any, against the mapping file.
- Mapping file against the geometry.

To check the active part number:

1. Read the required symbol (unless previously read into the session).
2. Read the required geometry (unless previously read into the session).
3. Choose the **Check > Part Number > Part Numbers...** pulldown menu item.

The Check Part Numbers dialog box opens.

4. Complete the dialog box as follows:
 - **Part Number**—contains the name of the active part number.
 - **Logic Symbol**—enter the name of a logic symbol to use in the check. This information directs LIBRARIAN to check the part number against a logic symbol other than the symbol associated with the part number.
 - **Check All in Active Catalog**—for this procedure, do not press this button.
 - **Check Against All Active Symbols**—press this button to check the part number against all available symbols with matching Comp property values.

Creating a Design Catalog

The design catalog is a custom catalog containing all the part numbers needed for a specific design. A complete design catalog ensures that all part numbers necessary for the design are present in the catalog. The complete design catalog also contains the mapping files for your design.

A complete and checked design catalog facilitates the packaging process in the PACKAGE tool. In PACKAGE, you need to load only one catalog file, the design catalog. The design catalog resides in the *design_maps* directory under your design directory. The name of the design catalog file is *design.catalog*.

Two menu items help you create the design catalog.

- **Check > Catalogs** compares the symbols in your design against the part numbers in the design catalog, and reports which symbols are missing part numbers in the design catalog.
- **Catalogs > Complete Catalog** searches for required part numbers in catalogs currently loaded in the LIBRARIAN session, then copies the part numbers and their mapping files into the design catalog. The search is based on the following:
 - a. LIBRARIAN looks primarily at the Comp property to match a symbol and a part number. If a symbol has a Part_no property and/or a Geom property, the property values must match the Part_no and/or Geom values in the part number description.
 - b. If the symbol has only the Comp property attached, you can specify additional properties to use for matching part numbers by including the Assign_prop statement in the *pkgconf* design object.

Lab Exercise

In this lab exercise you build the portion of your library that contains mapping and catalog files. You create several part numbers and mapping files to examine the options and capabilities of mapping file information.

Upon completion of this lab exercise you can:

- Create a catalog directory.
- Create a catalog.
- Add a part number to a catalog file.
- Build the logic description for a schematic symbol.
- Map the symbol to the component geometry.
- Check the catalog and mapping data.

Turn to Module 3—Lab 5: "Mapping Catalog Files".

Lab 5

Mapping and Catalog Files

Introduction

In this lab exercise you build the portion of your library that contains mapping and catalog files. You create several part numbers and mapping files to examine the options and capabilities of mapping file information.

Upon completion of this lab exercise you can:

- Create a catalog directory.
- Create a catalog.
- Add a part number to a catalog file.
- Build the logic description for a schematic symbol.
- Map the symbol to the component geometry.
- Check the catalog and mapping data.

Procedure

In this procedure, you invoke LIBRARIAN to create catalog and mapping data.

Preparation for Lab

You create catalog and mapping data in LIBRARIAN. To create catalog and mapping data you either 1. enter the schematic symbol data or 2. LIBRARIAN retrieves the data from schematic symbols created with the Mentor Graphics Symbol Editor. In this training you are not using data from Mentor Graphics generated symbol data, so you need to enter the symbol data. In this lab, you create part numbers for the symbols: 74act04 and 74ls373.

For each new part number you perform the following steps:

1. Read in the geometries.
2. Activate or create a catalog.
3. Create a part number in the active catalog.
4. Define the logical pin names for the schematic symbol definition of the map.
5. Map the symbol pins to the component geometry.
6. Repeat steps 3 through 5 for each new part number in the active catalog.
7. Check and save the catalog.

Now you can begin the lab:

1. Log in to your workstation.
2. Invoke LIBRARIAN from the Design Manager without the design (Stand Alone).

Reading in Geometries for 74act04 and 74ls373

The component geometry for a 74act04 is the soic14, and for the 74ls373 the geometry is the soic20. These components use the s80X25 and s80X25right padstacks. Now you read in these geometries.

1. Choose the **Geometries > List Geometry Libraries...** menu item. In the dialog box, view the User libraries, and then view the mgc.trng.components library. From the mgc.trng.components library, select and read the soic14 and soic20 components.

The geometries are read, and two geometry edit windows are displayed in the LIBRARIAN session. A report window might also be displayed, if so you can delete it.

2. Choose the **Geometries > List Geometry Libraries...** menu item again. View the User libraries, and then view the mgc.trng.padstacks library. From the mgc.trng.padstacks library, select and read the s80X25 and s80X25right padstacks.

The padstack geometries are read and displayed in edit windows.

3. Close the edit windows containing the s80X25 and s80X25right padstacks.

You can close the edit windows, because you do not need to edit the padstacks. You read the padstack geometries into the LIBRARIAN session so the soic14 and soic20 geometries can display the correct padstacks. Now that LIBRARIAN has the padstack information, it no longer needs the edit windows.

4. Choose the **Geometries > Visible Geometries Window...** menu item. In the dialog box containing a list of all open windows, choose the **CO\$soic14** window, and **OK** the dialog box.

The selected window is moved to the top of the window stack so you can view it.

Creating a Catalog

1. Choose the **Catalogs > Create Catalog:** menu item.

A prompt bar appears at the bottom of the Session window.

2. Fill in the prompt bar as follows, and then **OK** the prompt bar.

Name: **trng_maps** In Directory: **user**

This creates a catalog directory named *trng_maps*, with a catalog file called *trng.catalog* in the user directory. The catalog becomes the active catalog, and any part numbers that you create are placed in this catalog.

Creating a Part Number for 74act04

Now you create a part number in the active catalog that you made in the previous procedure.

1. Choose the **Catalogs > Create Part Number...** menu item.

The Create Part Number dialog box appears.

2. Fill in the dialog box with the following information, and then **OK** the dialog box.

Part number: **BHR-2X2705-40**
Catalog name: **trng_maps**
Geometry: **soic14**
Mapping: **pn-74act04-soic.map**
Directory: **User**

This is a *made up* part number. You normally use your standard company stock number or order number in the part number box. You would choose **Replace Part** only if the part number already exists in the catalog.

The geometry and mapping file are displayed on the screen. Notice that the mapping file window is blank. You define the symbol information in the next step.

Defining the Logic Symbol Data

This procedure assumes there is no logic symbol data in the design for the component, and you manually add logic pins.

1. Place the cursor in the map window, and then choose the **[Top Menu] Map Logic Symbol...** menu item.

The Map Logic Symbol dialog box is displayed.

2. Fill in the dialog box with the following information, and then **OK** the dialog box.

Select button:	Comp Property
Property Value:	74ACT04
Map Symbol Count:	Specify , then enter:
Map Symbol Count:	6
Map Symbol Name:	A
Swap Code:	1

The Map Logic Symbol dialog box defines six gates with no pins.

3. Place the cursor in the map window, and choose the **[Top Menu] Add Logic Pins...** menu item.

The Add Mapping File Pins dialog box appears.

4. Fill in the dialog box with the following information, and then **OK** the dialog box.

Create pin name: **I1**
Select button: **All Symbols**
Swap Code: **0**

Create pin name: **O1**
Select button: **All Symbols**
Swap Code: **0**

The Add Mapping File Pins dialog box defines the logic pins for each gate.

After you OK the dialog box, the pin information is added to the symbols in the map window. The physical pins (PHY) are not defined yet, so they are labeled *NONE*.

5. With the cursor in the map window, choose the **[Top Menu] Add Power Pins...** menu item.

The Add Mapping File Power Pins dialog box appears.

6. Fill in the dialog box with the following information, and then **OK** the dialog box.

Create pin name: **VCC**
Create pin name: **ground**

The Add Mapping File Power Pins dialog box defines the power pins for the component.

Mapping Logic Pins to Component Pins

1. Activate the edit window, then set up the grid to .05 with a display interval of 1.

This makes it easier for you to place the cursor on the pins of the geometry when you map logical pins to physical pins on the component.

2. With the cursor in the map window, choose the **[Top Menu] Map Pins:** menu item.

The Map Pins prompt bar appears.

3. Change the prompt bar option to repeat if it is not set to repeat. If you changed the repeat option, click the Select mouse button on Logic Pin Location, or press the TAB key to highlight the Logic Pin Location prompt.

4. Move the cursor to Symbol A, Pin: I1 (Intput 1) in the mapping file and click the Select mouse button. Observe that the line highlights.

The Select Physical Pin prompt bar opens.

5. Move the cursor to pin 1 of the soic14 geometry (in the geometry window) and click the Select mouse button. Observe that the mapping file now shows PIN I1 of Symbol A as PHYsical pin 1.

The Map Pins prompt bar returns.

6. Move the cursor to Symbol A, Pin: O1 (Output 1) in the mapping file, and click the Select mouse button. Observe that the line highlights.

The Select Physical Pin prompt bar reappears.

7. Move the cursor to pin 2 of the soic14 geometry and click the Select mouse button. Observe that the mapping file now shows PIN O1 of Symbol A as PHYsical pin 2.

The Map Pins prompt bar repeats.

8. Continue this process for each pin of the other symbols using the pin information below to map the 74act04 gates to the soic14 component geometry. Also map the two power pins with the same process.

SYMBOL A:
PIN "I1" "1"
PIN "O1" "2"

SYMBOL B:
PIN "I1" "3"
PIN "O1" "4"

SYMBOL C:
PIN "I1" "5"
PIN "O1" "6"

POWER "ground" "7"

SYMBOL D:
PIN "I1" "9"
PIN "O1" "8"

SYMBOL E:
PIN "I1" "11"
PIN "O1" "10"

SYMBOL F:
PIN "I1" "13"
PIN "O1" "12"

POWER "VCC" "14"

9. Cancel the prompt bar to end the Map Pin function.

Creating a Part Number for 74LS373

Now you repeat what you did for the 74act04 for the 74ls373.

Choose the **Catalogs > Create Part Number...** menu item. Fill in the Create Part Number dialog box with the following information, and then OK the dialog box.

Part number: **BHR-3X5250-39**
Catalog name: **trng_maps**
Geometry: **soic20**
Mapping: **pn-74ls373-soic.map**

The edit and map windows update with the correct model.

Defining the Logic Symbol Data

In this procedure you read the logic pin data from a logic symbol in a symbol library. This is the procedure you use most often when mapping pins.

1. Choose the **Symbols > Add Library Link** menu item. Fill in the dialog box as follows, then **OK** the dialog box.

Library Name: **sig_az.parts**

Pathname to Existing Symbol Library:

your_path/**training/board_new/mod3/parts**

Add to: **User**

Directory Type: **Permanent**

Remember, *your_path* is whatever pathname you have used to locate your **training** directory.

2. Choose the **Symbols > List Symbol Libraries** menu item. View the User library, and then view the sig_az.parts library. From the list of symbols, choose the **t1l373**, then read it.

After a moment, the dialog box list changes to show that t1l373 is a valid symbol, and that it has 18 pins. Also, the transcript in the message window shows a note stating that one default symbol was found for t1l373.

3. Close the dialog box.

4. Place the cursor in the map window, and choose the **[Top Menu] Map Logic Symbol...** menu item. Fill in the Mapping Logic Symbol dialog box with the following information, and then **OK** the dialog box.

Logic Symbol

Logic Symbol Name: **ttl373**

Map Symbol Count: **1**

Map Symbol Name: **A**

You cannot enter a swap code for the symbol because there is only one symbol.

The map window changes to display the logical pin to physical pin mapping.

5. From the map window, choose the **[Top Menu] View Logic Symbol** menu item.

A new window displays below the map window showing the logic symbol so you can compare it with the physical geometry.

6. Close the logic symbol window when you finish viewing the logic symbol.
7. In the map window, choose the **[Top Menu] Add Power Pins...** menu item. Fill in the Add Mapping File Power Pins dialog box with the following information, and then **OK** the dialog box.

Create pin name: **VCC**

Create pin name: **ground**

Mapping Logic Pins to Component Pins

1. In the map window, choose the **[Top Menu] Map Pins:** menu item.

The Map Pins prompt bar appears.

2. If the prompt bar repeat option is not set, change the prompt bar option to repeat. Then, click the Select mouse button on Logic Pin Location, or press the TAB key to highlight the Logic Pin Location prompt.
3. Assign the VCC pin to physical pin 20 by clicking on VCC in the map window, and then clicking on physical pin 20 in the geometry window. Assign the ground pin to pin 10 using the same method. Use the information below to verify the mapping of the 74ls373 pins to the soic20 component geometry. If any of the logic pins in the map window are unassigned (state NONE in the map window), select that pin in the map window, and then click on the correct pin in the geometry window according to the following list.

SYMBOL A:

PIN "D1" "3" 0
 PIN "D2" "4" 0
 PIN "D3" "7" 0
 PIN "D4" "8" 0
 PIN "D5" "13" 0
 PIN "D6" "14" 0
 PIN "D7" "17" 0
 PIN "D8" "18" 0

POWER "VCC" "20"

PIN "Q1" "2" 0
 PIN "Q2" "5" 0
 PIN "Q3" "6" 0
 PIN "Q4" "9" 0
 PIN "Q5" "12" 0
 PIN "Q6" "15" 0
 PIN "Q7" "16" 0
 PIN "Q8" "19" 0
 PIN "OC" "1" 0
 PIN "C" "11" 0

POWER "ground" "10"

If you accidentally add the wrong physical pin to a logical pin, just select the logical pin again in the map window, and then select the correct physical pin.

4. Cancel the prompt bar to end the Map Pins function.

Defining Pin Sets

1. In the map window, choose the **[Top Menu] Make Pinset...** menu item. Fill in the Make Pinset dialog box with the following information, and then **OK** the dialog box.

Select button: **Comp Property**
t1l373
 Pinset name: **ps1**
 Pin names: **D1 Q1**
 Swap Code: **1**

2. Repeat for each of the following pinsets:

Comp Property

Property Value: **t1l373**
 Pinset name: **ps2**
 Pin names: **D2 Q2**
 Swap Code: **1**

Comp Property

Property Value: **t1l373**
 Pinset name: **ps3**
 Pin names: **D3 Q3**
 Swap Code: **1**

Comp Property

Property Value: **t1l373**
 Pinset name: **ps4**
 Pin names: **D4 Q4**
 Swap Code: **1**

Comp Property

Property Value: **t1l373**
 Pinset name: **ps5**
 Pin names: **D5 Q5**
 Swap Code: **1**

Comp Property

Property Value: **t1l373**
 Pinset name: **ps6**
 Pin names: **D6 Q6**
 Swap Code: **1**

Comp Property

Property Value: **t1l373**
 Pinset name: **ps7**
 Pin names: **D7 Q7**
 Swap Code: **1**

Comp Property

Property Value: **t1l373**
 Pinset name: **ps8**
 Pin names: **D8 Q8**
 Swap Code: **1**

Checking and Saving Part Numbers and Mapping Files

When you complete mapping the two part numbers, you need to check the part numbers before you save them in the **trng_maps** catalog.

1. Check the catalog data you created by choosing the **Check > Catalogs...** menu item.

The Check Catalogs dialog box appears.

2. In the dialog box, enter the catalog name, **trng_maps**, and select User. **OK** the dialog box.

After you have executed the preceding menu command, a report window appears in the display. The message lists all the part numbers checked and any errors or warnings found. Correct the errors and verify the warnings before saving the catalog. You see no errors or warnings.

3. Close the report window.
4. If the part numbers check without error, save the catalog data by choosing the **File > Save > Design Specify...** menu item. In the Save Design Specify dialog box, select Catalogs from the *Save to design:* field, and **OK** the dialog box.

The **trng_maps** container (directory) you just wrote is saved in your **pcb_parts/user_maps** directory.

5. Close the LIBRARIAN session.

If you are prompted to save the design, choose No.

Congratulations! You have completed the *Creating PCB Design Geometries* module. The next module of the *Board Station for New Users Training Series* is Module 4: *Packaging the Design for LAYOUT*.

INDEX

A

- Active library 5-26, 5-28
- Add
 - logic pins 5-41
 - pins 3-17
- Add Mapping File Pins dialog box 5-41
- Add Mapping File Power Pins dialog box 5-35
- Attributes
 - adding pins 3-17
 - Artwork_void 4-8
 - Board_default_padstack 4-5
 - Board_external_copper 4-5
 - Board_internal_copper 4-5
 - Board_material 4-5
 - Board_placement_grid 4-5
 - Board_placement_keepout 4-9
 - Board_placement_outline 4-9
 - Board_placement_region 4-9
 - Board_routing_layers 4-5
 - Board_routing_outline 4-8
 - Board_thickness 4-5
 - Component_default_padstack 3-12
 - Component_diagonal_allowed 3-13
 - Component_edge_connector 3-12
 - Component_height 3-12
 - Component_layout_surface 3-14
 - Component_layout_type 3-14
 - Component_not_in_bom 3-12
 - Component_orthogonal_only 3-13
 - Component_outline_overhang 3-12
 - Component_padstack_override 3-16
 - Component_pin_definition 3-16
 - Component_pins_moveable 3-12
 - Component_placement_outline 3-14
 - Component_specific_layer_off 3-14
 - Component_specific_layer_on 3-14
 - Component_thermal_outline 3-14
 - components 3-13
 - Default_pad_size 4-5

- Diagonal_placement_allowed 4-6
- Diagonal_routing_allowed 4-6
- Mechanical_parts 3-16, 4-9
- Orthogonal_placement_only 4-6
- pin padstacks 3-9
- Power_net_names 4-7
- Routing_keepout 3-14, 4-9
- Terminal_2_layer_definition 3-10
- Terminal_blind_definition 3-8
- Terminal_buried_definition 3-10
- Terminal_drill_size 3-8, 3-10
- Terminal_surface_definition 3-8
- Terminal_thruhole_definition 3-8
- Terminal_thruvia_definition 3-10
- Tjunctions_allowed 4-6
- Trace_keepout 3-14, 4-9
- via padstacks 3-11
- Via_keepout 3-14, 4-9

B

- Board
 - geometry type 1-7

C

- Catalog files
 - definition 5-3
 - example 5-17, 5-19
- Catalogs > Create Catalog 5-48
- Catalogs > Create Part Number 5-48, 5-52
- Categories
 - of libraries 1-4, 5-3
- Change Geometry > Add Padstack Overrides 3-38
- Change Geometry > Add Placement Outline 3-31, 4-34
- Change Geometry > Add Routing Outline 4-36
- Check > Catalogs 5-57
- Check > Geometry > Active Geometry 3-32

INDEX [continued]

Company Library

defined 1-22

Components

adding pins to 3-17

defined 1-3

geometry type 1-6

Container

pcb structure 1-20

Copper clearances by layer

defined 4-6

Crop marks 4-10

D

Default

physical layers 4-13

Default Catalog Library Hierarchy dialog box
5-24

Default Geometry Library Hierarchy dialog
box 1-25

Design Library

defined 1-22

Design objects

location for pcb 1-20

Dialog boxes

Add Mapping File Pins 5-41

Add Mapping File Power Pins 5-35

Default Catalog Library Hierarchy 5-24

Default Geometry Library Hierarchy 1-25

Map Logic Symbol 5-33

Part Number List 5-26, 5-28

Setup Physical Layers 4-12

Directory hierarchy

geometry library pathnames 1-23

Drill hole clearance

defined 4-6

E

Examples

catalog files 5-17, 5-19

mapping files 5-9, 5-38

F

File > Save > ASCII Geometries 1-54, 1-55, 3-49, 3-50, 4-53

File > Save > Design 4-54

File > Save > Design > Catalogs 5-57

G

Geometries

board 1-7

changing generic 3-6

component 1-6

defined 1-3

via padstack 1-5

Geometries > Add Library Link 1-46

Geometries > Create Geometry > Board 4-19

Geometries > Create Geometry > Buried Via 3-24

Geometries > Create Geometry > Component
3-25, 3-33, 3-36, 3-43

Geometries > Create Geometry > Surface Pin
3-23

Geometries > Create Geometry > Thruhole Pin
3-21

Geometries > List Geometry Libraries 1-47, 4-39, 5-47

INDEX [continued]

L

- Layer rules
 - defined 4-11
- Layers
 - Board_outline 1-11
 - Breakout 1-15
 - Dam 1-11
 - Density 1-10
 - Drawing 1-15
 - Drill 1-11
 - Drill_holes 1-11
 - Errors 1-10
 - Glue_mask 1-15
 - Milling 1-11
 - Off_grid_pins 1-10
 - Pad 1-15
 - Panel_outline 1-12
 - Paste_mask 1-15
 - Pin_id 1-16
 - Place 1-16
 - Placement_keepout 1-12
 - Placement_region 1-12
 - Power 1-13
 - Route_grid 1-10
 - Routing_keepout 1-12
 - Signal 1-14
 - Silkscreen 1-16
 - Solder_mask 1-16
 - Thermal 1-17
 - Trace_keepout 1-13
 - Via 1-14
 - Via_keepout 1-13
 - Via_usage 1-13
- Libraries
 - categories of 1-4, 5-3
- Logos 4-10

M

- Map > Add Logic Pins 5-49
- Map > Add Power Pins 5-50, 5-54
- Map > Make Pinsets 5-56
- Map > Map Logic Symbol 5-49, 5-54
- Map > Map Pins 5-51, 5-55
- Map Logic Symbol dialog box 5-33
- Mapping
 - with a logic symbol 5-33
 - without a logic symbol 5-39
- Mapping files
 - adding pins 5-41
 - adding power pins 5-35
 - common statement definition 5-9
 - create with logic symbol 5-33
 - create without logic symbol 5-39
 - definition 5-3
 - example, common pin 5-9
 - example, complete 5-38
 - mapping pins 5-36
 - setting swap codes 5-37
 - swap codes 5-36
- Mapping windows
 - configuration 5-31
- Mentor Graphics Library
 - defined 1-22

N

- Net rules
 - defined 4-11
- Net rules for layers
 - defined 4-11

INDEX [continued]

P

Part Number List dialog box 5-26, 5-28

Parts

defined 1-3

Physical layer rules

default layers 4-13

defined 4-10

Pin padstacks

geometry type 1-5

Pin rules

defined 4-10

Pins > Add Pin 3-26

Pins > Add Pins Array 3-37, 3-44, 3-45

placement outline clearance

defined 4-6

Project Library

defined 1-22

R

Route

defined 1-3

Routing outline clearance

defined 4-6

S

Setup > Grid 1-35, 1-52

Setup > Line Width 1-35, 3-29

Setup > Snap Direction 1-43

Setup > Text 1-41, 3-30

Setup Design Rules > Layer Rules 4-45

Setup Design Rules > Net Rules 4-49

Setup Design Rules > Physical Layers 4-42

Setup Physical Layers dialog box 4-12

Shapes > Add Arc > Point Angles 1-38

Shapes > Add Arc > Radius Angles 1-36

Shapes > Add Circle > Center Points 1-44

Shapes > Add Line > Add Line 1-39, 3-29

Shapes > Add Polygon 1-43

Shapes > Copy > Copy 1-38

Shapes > Extended Menu > Add Drill Hole 4-32

Shapes > Extended Menu > Add Geometry > Add Geometry 4-39

Shapes > Flip > Vertically 1-39

Shapes > Move 1-53

Shapes > Move > Origin 3-48

Shapes > Unselect 1-39

T

Targets 4-10

Text > Add Interactive Text 1-41

Text > Add Reference Name 3-30

Traces

defined 1-3

U

User Library

defined 1-22

V

Via padstacks

geometry type 1-5

Via rules

defined 4-10

View > Change View Style 1-42, 1-45

W

Wires

defined 1-3

欢 迎 光 临

微波 EDA

<http://www.mweda.com>

EDA学习网

<http://edastudy.ik8.com>

<http://www.mweda.com>

<http://edastudy.ik8.com>

本站专业提供各种微波仿真软件和 PCB 设计软件的安装光盘和学习培训教程，本站提供的所有教程教材都是公司培训用的，很系统的讲述了相关软件的应用。主要项目有：

破解软件类：微波仿真软件

ADS2004A	ADS2003C	ADS2003A		
HFSS9.2	HFSS9.1	HFSS9.0	HFSS8.0	
Ansoft Designer1.1	Ansoft Serenade8.71	Ansoft Maxwell 10	Ansoft SIWave	
Microwave Office2002	Sonnet Suite Pro 9.52	CST5.0		
Super NEC 2.5	Zeland IE3D9.2	XFDTD 6.0		

破解软件类：PCB 工具软件

Mentor EN2004	Mentor EN2002	Mentor ePD2004	Mentor SDD2004
Mentor WG2004	Mentor WG2002	Mentor ISD2004	
PowerPCB5.0	PowerLogic5.0	Orcad10.3	
PADS2005	PADS2004		
Cadence SPB15.2 (Allegro 15.2)		Cadence PSD15.0 (Allegro 15.0)	

软件学习、培训教程：

Mentor EN: Mentor EN 原版培训教程
Mentor EN 视频教程

Mentor WG: Mentor Expedition / Mentor WG 中文用户手册
Mentor Expedition (WG) PCB Training Workbook
Mentor DxDesigner Design Processing Training Workbook

HFSS : HFSS 9.2 入门与提高教程
HFSS 9.0 入门与提高教程
HFSS9 视频教程
HFSS9 Training Tutorials
HFSS8 Training Manual
电子科大 HFSS8 教程

ADS: ADS 中文基础教程
ADS 设计实验教程
ADS 入门教程
ADS2003 Training Workbook: ADS2003 fundamental
ADS Training Workbook for Momentum
ADS Customization Training Workbook

Using ADS Communication Systems Designer

Using ADS to Design WCDMA/3GPP Communication Systems

Ansoft Designer:

Ansoft Designer 2.1 Full Book

Ansoft Designer System Training Guide

Ansoft Designer 1.1 入门与提高

Ansoft Designer 多媒体教程

Ansoft Designer Training 讲义

Cadence Allegro:

Allegro 15.2 原版教程

Concept-HDL 15.2 原版教程

Allegro 14.2 原版教程

Allegro 视频教程

Allegro PCB Layout 高速电路板设计

注：本站还有Ansoft Maxwell, Ansoft Serenade, Ansoft Q3D , Zeland IE3D, PowerPCB , Orcad等的培训教程。详情可登陆：<http://www.mweda.com> 或 <http://edastudy.ik8.com>

欢 迎 光 临

微波 EDA

<http://www.mweda.com>

EDA学习网

<http://edastudy.ik8.com>

<http://www.mweda.com>

<http://edastudy.ik8.com>

本站专业提供各种微波仿真软件和 PCB 设计软件的安装光盘和学习培训教程，本站提供的所有教程教材都是公司培训用的，很系统的讲述了相关软件的应用。主要项目有：

破解软件类：微波仿真软件

ADS2004A	ADS2003C	ADS2003A		
HFSS9.2	HFSS9.1	HFSS9.0	HFSS8.0	
Ansoft Designer1.1	Ansoft Serenade8.71	Ansoft Maxwell 10	Ansoft SIWave	
Microwave Office2002	Sonnet Suite Pro 9.52	CST5.0		
Super NEC 2.5	Zeland IE3D9.2	XFDTD 6.0		

破解软件类：PCB 工具软件

Mentor EN2004	Mentor EN2002	Mentor ePD2004	Mentor SDD2004
Mentor WG2004	Mentor WG2002	Mentor ISD2004	
PowerPCB5.0	PowerLogic5.0	Orcad10.3	
PADS2005	PADS2004		
Cadence SPB15.2 (Allegro 15.2)		Cadence PSD15.0 (Allegro 15.0)	

软件学习、培训教程：

Mentor EN: Mentor EN 原版培训教程
Mentor EN 视频教程

Mentor WG: Mentor Expedition / Mentor WG 中文用户手册
Mentor Expedition (WG) PCB Training Workbook
Mentor DxDesigner Design Processing Training Workbook

HFSS : HFSS 9.2 入门与提高教程
HFSS 9.0 入门与提高教程
HFSS9 视频教程
HFSS9 Training Tutorials
HFSS8 Training Manual
电子科大 HFSS8 教程

ADS: ADS 中文基础教程
ADS 设计实验教程
ADS 入门教程
ADS2003 Training Workbook: ADS2003 fundamental
ADS Training Workbook for Momentum
ADS Customization Training Workbook

Using ADS Communication Systems Designer

Using ADS to Design WCDMA/3GPP Communication Systems

Ansoft Designer:

Ansoft Designer 2.1 Full Book

Ansoft Designer System Training Guide

Ansoft Designer 1.1 入门与提高

Ansoft Designer 多媒体教程

Ansoft Designer Training 讲义

Cadence Allegro:

Allegro 15.2 原版教程

Concept-HDL 15.2 原版教程

Allegro 14.2 原版教程

Allegro 视频教程

Allegro PCB Layout 高速电路板设计

注：本站还有Ansoft Maxwell, Ansoft Serenade, Ansoft Q3D , Zeland IE3D, PowerPCB , Orcad等的培训教程。以及射频/微波/天线类英文原版书籍。详情可登陆：

<http://www.mweda.com> 或 <http://edastudy.ik8.com>